Design and Optimization of Lug Bracket Assembly

G. GOWTHAM*[,1], G. SHIVA SAM KUMAR[1], AASA DARA[1]

*Corresponding author
[1]Department of Aeronautical Engineering,
Veltech Rangarajan Dr. Sagunthala R&D Institute of Science & Technology,
Chennai, India
gowthamg@veltech.edu.in*, shivasamkumar1@gmail.com, aasad3698@gmail.com

DOI: 10.13111/2066-8201.2021.13.1.6

Received: 17 June 2020/ Accepted: 10 January 2021/ Published: March 2021

Abstract: *An aircraft is an advanced mechanical structure made by man which has been dominating the skies from the early 19th centuries. It has been used for transportation of cargo/ passengers from one place to another in a shorter period of time. Advances in aeronautics lead to the development of fighter aircrafts with exciting and dominating characteristics. A fighter aircraft is to be designed in such a way that it can withstand heavy loadings on the wing due to its high manoeuvrability. A fighter aircraft is designed to be marginally unstable, which makes control easier and better during manoeuvrability at high speeds, but in this state there is a heavy fluctuating load acting on the wing. The wing is connected to the fuselage using wing fuselage lug attachment bracket. Since the wing is a cantilever structure, the load acting on the wing is concentrated on the hinge (lug bracket assembly). In this paper, a lug bracket is designed according to the standard design procedure and is validated using Finite Element Methods to ensure the static loading capability and stress concentrations in lug bracket. The validated model has been optimized using Altair Optistruct. The optimized model has been validated under static loading condition for the stress concentration and displacement and is compared with initial model in order to study and understand its behaviour under various conditions.*

Key Words: *Fighter aircraft, Finite element method, optimization, stress concentration, static loading condition, validation*

1. INTRODUCTION

Aircraft are the most popular transportation in recent times as they reduce travel time. They are used to transport goods and civilians and are also used for military purposes. Their size, shape and configurations differ depending on the requirements. It is rather a challenging task to design a fighter aircraft, as it dominates the sky through its high speed and must withstand many loads and difficulties due to manoeuvrability and be able to accomplish the task without affecting any structure [1], [5]. The structure of an aircraft is mainly divided into three basic parts such as fuselage, wing and empennage.

Wing and fuselage are considered important parts of an aircraft. As wings are subjected to different loads, they are supposed to be rigidly fixed to the fuselage. This attachment is done by a series of pinned lugs between the wing side of wing box and the fuselage through which the bending moment and shear loads are transferred from wing to fuselage thus, aircraft wing-fuselage lug attachment bracket is the one on which the maximum loads act [3], [9]. When the lug undergoes a catastrophic failure, it may lead to the separation of the entire aircraft structure.

One end of the lug is attached to the fuselage and the other end is attached to the I-spar of the wing. Hence, the entire structure of the wing acts as a cantilever beam with the lug bracket attached/ fixed to the fuselage which transfers the load to the fuselage. The connection between the wing and the fuselage of the aircraft occurs with four lugs of which, two at the front spars and two at the rear spars.

The entire load is evenly distributed among these spars and is transferred through the lug pin. In order to increase the lifetime of this lug bracket and to reduce the frequent inspection cost, Finite Element Analysis (FEA) is performed to validate the stress concentrations and optimize the lug-bracket assembly (Topology and shape optimisation) for easy manufacturability with minimized constraints [2], [4].

2. OBJECTIVE

The objectives of this project are as follows:

- To design a new model according to the standard design procedure and the reference lug models.
- To conduct linear static analysis of the bracket to obtain:
  - Maximum von-Mises stress
  - Maximum shear stress
  - Other necessary contours
- To perform modal analysis of the lug attachment bracket to analyse the behaviour of the lug under the natural frequency.
- To interpret the results of linear static and conduct optimization according to topology and shape to reduce or constrain the values of displacement stress and frequency values.
- To derive a final model which has less volume fraction, weight, minimum compliance and reduced stress and displacement contours which can be replaced with the existing model.

3. DESIGN TERMINOLOGY

Here we re-create a new model for lug-bracket assembly. To do that, first we have to decide how the new lug model should look like. The pre-existing lug models from references [16], [20] are obtained and studied.

The lug has to be designed with the dimensions and geometry that will help the lug bracket to withstand the heavy loading conditions derived in the previous section. The dimension for the lug bracket is achieved through the standard design procedure which is as follows. The material we have used is Steel AISI-4340.

The load applied on the lug-bracket assembly is 90584.62 N (P). [12], [16], [20] for light weight fighter aircraft.

Here, the design is based on yield stress i.e., σ_{yt} = 1550 N/mm^2 (which is the yield stress of the material used). Considering,

$$\sigma_{yt} = \frac{P}{A} \tag{1}$$

Substituting the yield stress σ_{yt} and load P in the above equation, we get the diameter of the pin hole D_p.

$$1550 = \frac{90584.62}{\pi r^2} \tag{2}$$

$$r = 24.795 \text{ mm}$$
$$D = 49.59 \text{ mm} \tag{3}$$

Bearing stress is calculated as

$$\sigma_{bearing} = \frac{P}{A_{bearing}} = \frac{P}{D \times t} \tag{4}$$

$$\text{Bearing strength} = 0.5 \times \ \sigma_{yt} = 0.5 \times 1550 = 775 \frac{\mathrm{N}}{\mathrm{mm}^2} \tag{5}$$

But,

$$\sigma_{bearing} = \frac{P}{D \times t} \tag{6}$$

Substituting load *P*, Diameter *D* and obtained bearing stress, we get the thickness t, of our lug.

$$775 = \frac{90584.62}{r \times t} \tag{7}$$

$$t = 80\text{mm (approx.)} \tag{8}$$

$$\text{Height of the lug hole} = 3.5 \times D_i = 181.35 \text{ mm} \tag{9}$$

A 3-D model of lug bracket with the above dimensions is modelled using CATIA-V5 (Designing tool) and drafted down to visualize the model in different projections as shown in fig. 1.

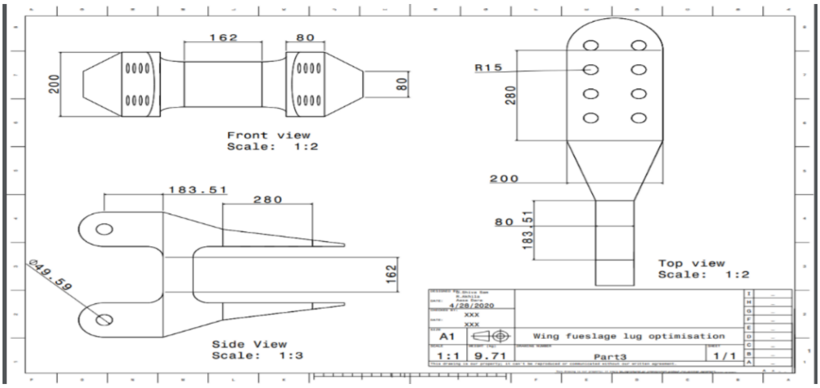

Fig. 1 – Orthographic projections

4. MATERIAL SELECTION

In Aeronautical industry, strength, rigidity and weight are the most important properties in material selection. In many situations trials and errors can be very expensive and for a good project design, material selection is very important. The material properties to be considered for the selection of material for structural applications are:

- Yield stress
- Ultimate Tensile stress
- Temperature limits
- Fatigue resistance
- Corrosion resistance

- Fracture toughness
- Crack growth resistance
- Ductility

In this paper AISI-4340 is used to model the lug bracket. The steel alloy is examined according to the material properties considered for selection in structural applications [22], [21]. The material properties are tabulated below in Table 1.

Table 1. – Material properties

PROPERTIES	STEEL AISI-4340
Young's Modulus, E	$211000 \frac{N}{mm^2}$
Poison's Ratio, μ	0.3
Ultimate Tensile Strength, UTS	$2200 \frac{N}{mm^2}$
Yield Stress, σ_y	$1550 \frac{N}{mm^2}$
Density, ρ	$7.85 \frac{g}{cm^3}$

5. FINITE ELEMENT ANALYSIS

Finite Element Method (FEM) is widely used for solving engineering problems and mathematical models. It is a particular numerical method used to solve partial differential equations consisting of two or three variables. It is increasingly becoming the primary tool for designers and analysts [1], [3]. The new model for lug-bracket assembly of fighter light-weight aircraft is recreated.

The structural behaviour of the component is identified and studied using a technique called Finite Element Analysis (FEA). The analytical solution for the lug bracket can be solved using FEA solutions such as Ansys, Nastran-Patran, Ansa, Hyperworks. In this project for linear static analysis and optimization we use Hyperworks [5], [7], [8]. Altair Hyperworks is the most comprehensive simulation platform offering the best set of solvers to design and optimize to high performance increasing the efficiency of the product.

6. DISCRETIZATION

Meshing also known as Discretization is a process of dividing an element into n-number of smaller elements. The accuracy in the analytical solution of the component highly depends on the quality of the mesh in the component [13], [14]. Meshing can be classified into two types based on one quality.

- Coarse mesh = medium sized elements
- Fine mesh = very small and fine sized elements

The time taken for solving the problem depends on the size of the elements, if the element size is smaller the solving time is longer [16], [17], [18]. Meshing based on the element can be classified as 1-D, 2-D and 3-D meshing.

In this paper we are using 3-D element for meshing the lug bracket. 3-D elements are generally used when all the dimensions are comparable. There are various types of 3-D elements such as Tetra, Penta, Hexa or Brick and Pyramid. 3-D elements are generally used to mesh solid mould components such as Gear box, Engine box, Crank shaft, etc [15]. A 3-D tetra element is used to mesh the lug bracket. Both the R-Tria and Tria elements are set as the base elements in the 3-D tetra mesh [19], [20]. The lug bracket had been divided into two parts, one is the design space and other is the non-design space.

The nodes across the bolt holes had been joined through 16 rigid body element (RBE2). A total of 37,588 elements are used in the non-design space and 89,752 elements are used in the design space part of the lug-bracket. The RBE2 elements are constrained to 6 degrees of freedom to transfer all the loads equally to pin holes, as shown in fig. 2.

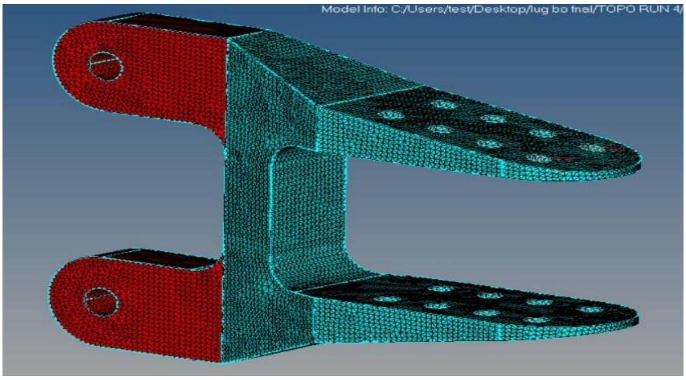

Fig. 2 – 3-D Tetra mesh

7. LOADS AND BOUNDARY CONDITIONS

The pin holes of the lug joint has been constrained to 6 degrees of freedom (yellow) and a total force of 5661.538 N [12], [16] is applied to each lug hole, since the end flange is to be attached to the I-spar; the lift load is represented in positive Y direction in the interconnected nodes of lug holes as shown in figs. (3-4).

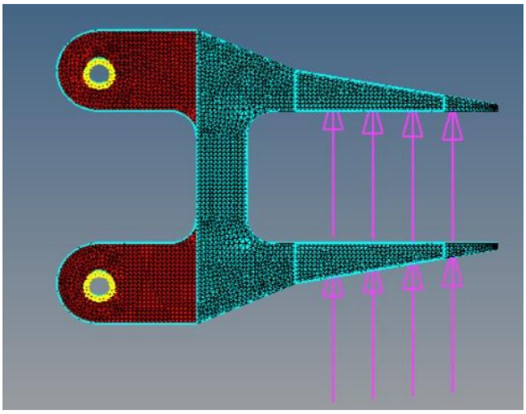

Fig. 3 –Fixed to 6 degrees of freedom

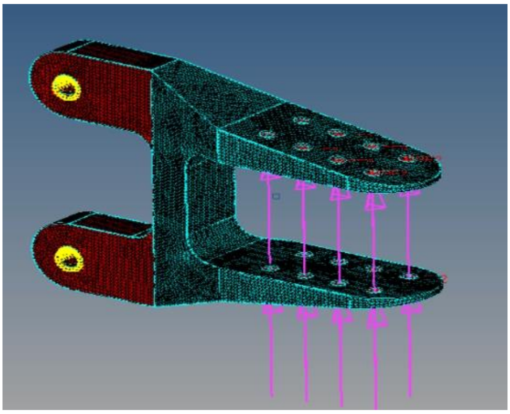

Fig. 4 – Loads distributed equally

8. RESULTS AND INTERPRETATION

After setting the loads and boundary conditions for the re-created lug-bracket it is ready for analysis. PARAM, SCREEN and OUTPUT Cards are used for the Analytical solution. The results for the shear stress, von-Mises stress, von-Mises strain and displacement are obtained. The contours for the results are obtained as shown in figs. (5-8).

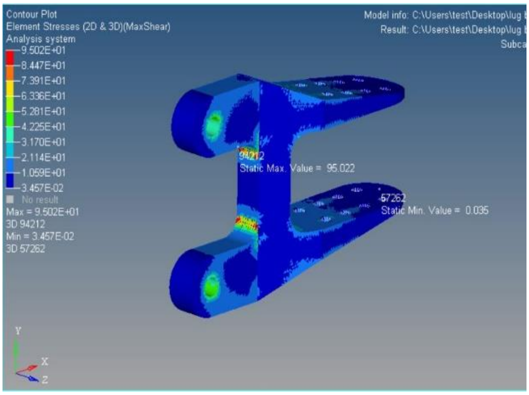

Fig. 5 – Shear stress

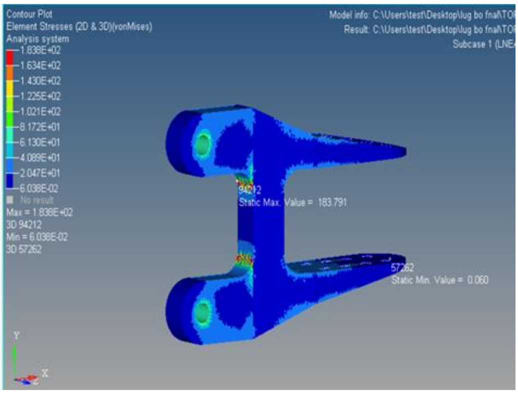

Fig. 6 – Von-Mises stress

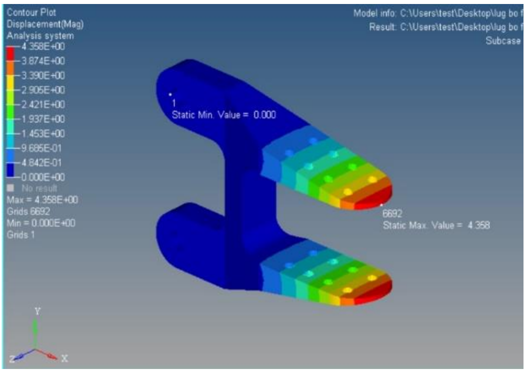

Fig. 7 – Displacement

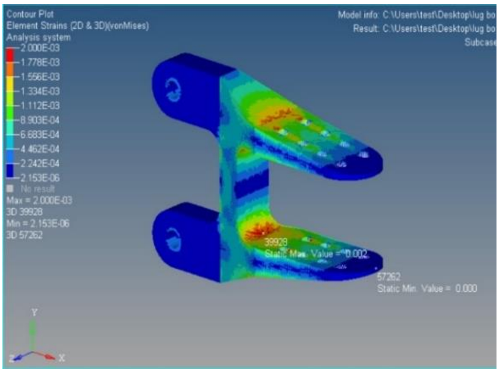

Fig. 8 – Von-Mises strain

The results for shear-stress, von-Mises stress, von-Mises strain and displacement are tabulated in Table 2 below and interpreted. From the contour plots it is identified that the maximum von-Mises stress is located in the fillet regions. This region may be the starting point of the crack and the volume of the lug bracket is too high [19]. The displacement values are higher at the flange tips which leads to a heavy vibration in the wings. To reduce the volume, the material orientation has to be identified and optimized [11].

Table 2. – Results

S. NO	CONTOURS	VALUE
1.	Maximum Von-Mises stress	183.791 MPa
2.	Maximum Shear stress	95.022 MPa
3.	Maximum Von-Mises strain	0.002
4.	Displacement	4.358 mm
5.	Weight	0.851 kg
6.	Volume	1.15 mm^3

9. OPTIMIZATION AND TYPES

Optimization in general is defined as the method of finding a best and satisfying solution from all the feasible solutions.

Structural optimization is a discipline or branch dealing with optimal design of load-carrying mechanical structures [3].

Given a pre-defined design domain (in two or three dimensions), external loads and material to be used are defined. The problem is to define an optimal structure to carry these loads. The objective of the optimization problem may be stated as reduced weight, constrained displacement and modal frequency values. The optimization can be classified into:

- Topology optimization,
- Shape optimization,
- Size optimization,
- Topography optimization.

10. TOPOLOGY OPTIMIZATION

Topology optimization is a mathematical method that optimizes the material layout within a given design space, for a given set of loads, boundary conditions and constraints with a goal of maximizing the performance of the system.

Topology optimization provides a perfect orientation for the material placement and shows us the places from which the material can be removed without affecting the stiffness and rigidity of the structure and reduces the volume of structure [2], [5].

In this section the lug-bracket is optimized to optimal shape using the topology optimization. Optimal shape for the lug bracket is obtained by removing the material from the area due to which there will be no effect in stiffness of the structure. The constraints for the topology optimization is defined in Table 3.

Table 3. – Topology Optimization constraints

Objective	Minimize volume Minimize weighted compliance
Response	Von-Mises stress < 180 MPa Weighted compliance Nodal displacement constrained in Y-axis
Constraints	Displacement < 1.5 mm Shear stress <95 MPa

The procedure for the optimization is defined using a formula

D = Design variable
R = Responses
D = De constraint
O = Objective

The four general steps to be followed for any optimization is given as follows.

- The design space is defined i.e. the place from where the material is to be removed is identified and selected.
- Responses are nothing but the definition of the objective and constraints.
- De-constraint panels offer the space to define the lower and upper bound for the constraints.
- Objective panel defines the final objective of the optimization like minimum volume, maximum frequency, minimum and maximum Von-Mises stress, etc.

The topology optimization following these steps is performed. The stress and nodal displacements are constrained as mentioned in the table.

The element density is obtained from the results of topology optimization as shown in Fig. 9.

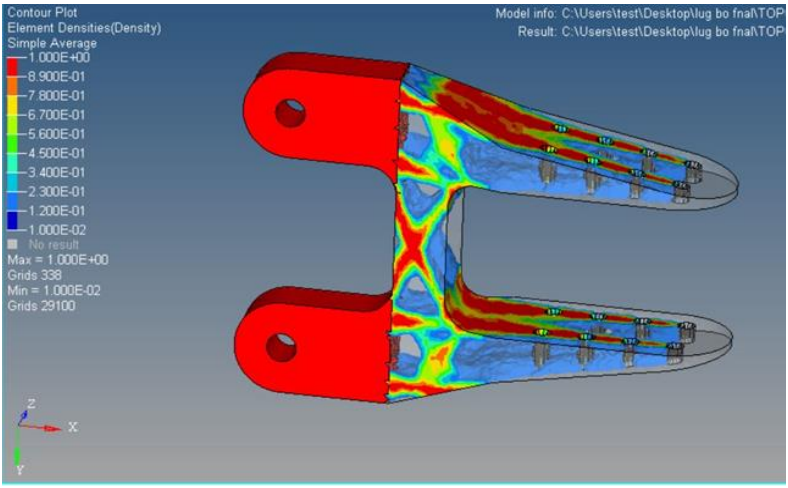

Fig. 9 – Material orientation

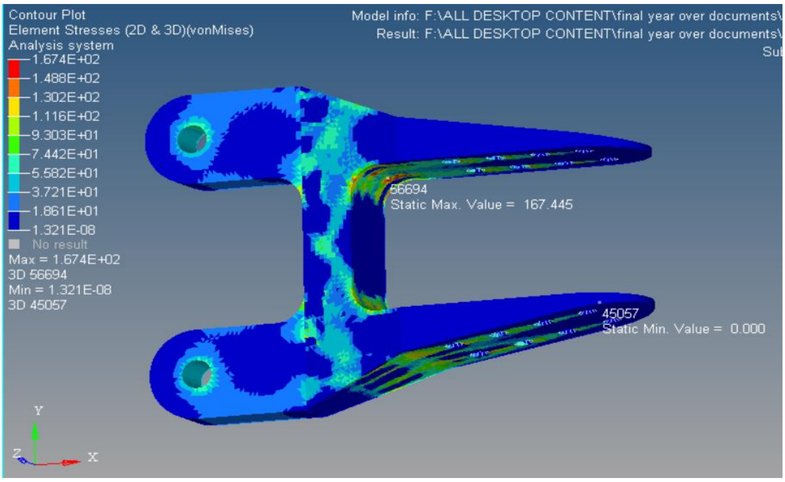

Fig. 10 – Von-Mises stress after 75th iteration

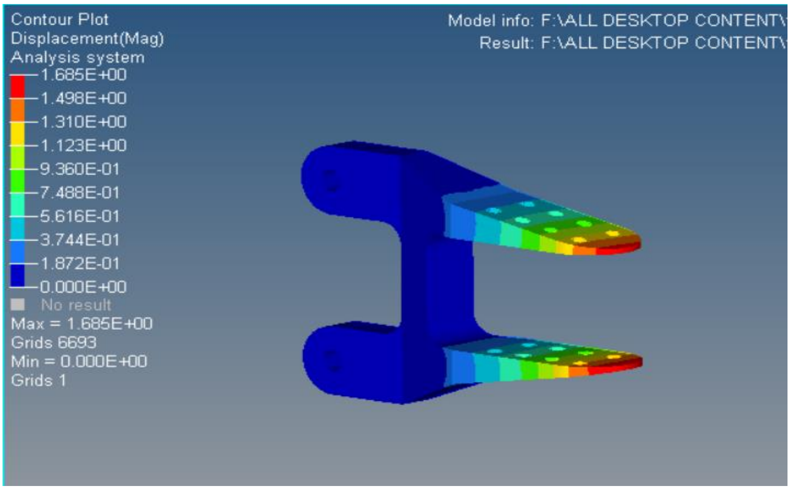

Fig. 11 – Displacement after 75th iteration

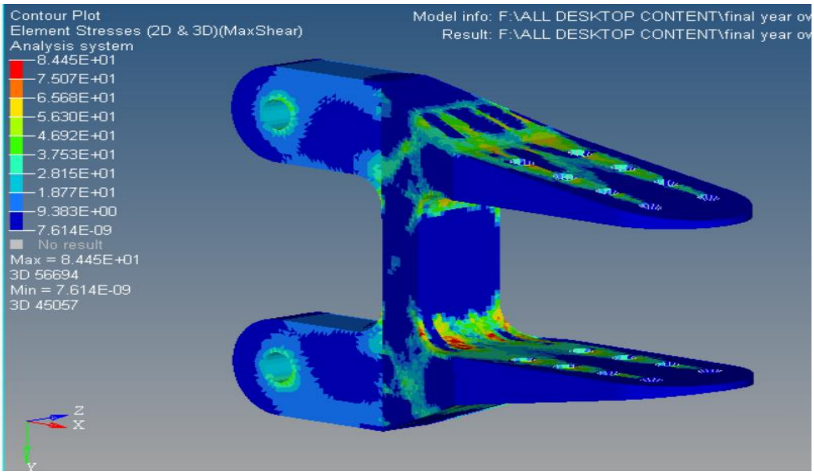

Fig. 12 – Shear stress after 75th iteration

The change in von-Mises stress, von-Mises strain and displacement after the iteration (optimization) is shown in Fig. (10-12). The results of optimization are tabulated in Table 4.

Table 4. – Topology Optimization results

Iteration	**Von-Mises stress (Mpa)**	**Von-Mises strain**	**Displacement (mm)**	**Max shear stress (Mpa)**	**Weight (kg)**	**Volume (mm^3)**
0th	183.791	0.02	4.358	95.022	0.851	1.15
75th	167.45	0.02	1.685	84.499	0.651	0.908

11. SHAPE OPTIMIZATION

In a structural shape optimization problem the aim is to improve the performance of the structure by modifying its boundaries.

This can be numerically achieved by minimizing an objective function subjected to certain constraints [17].

The typical problem of shape optimization is to find the shape which is optimal and minimize certain cost functions while satisfying the given constraints. In the previous section the lug-bracket was optimized using the topology optimization. In the Fig. 12 it is seen that the critical stress areas are at the fillets [19].

So in this section the fillets in the inner region of the lug bracket are optimized. The constraints for the shape optimization are as in Table 5.

Table 5. – Shape Optimization constraints

Objective	Minimize stress (maximum von-Mises stress)
Constraints	No constraints
Design variable	Grids move normal to the surface

The procedure for shape optimization is the same as for the topology optimization, but some changes are done in the design space and design variable because the mode of optimization is different.

In this section we will learn about the procedure followed for the free-shape optimization in Optistruct.

- The Hypermesh solver is opened and the Optistruct user profile is loaded.
- The solver setup for lug bracket is installed and the property collector, load collector, and the material collector are verified.
- A new component and property collectors are created for the elements in the fillet region.
- The elements in the fillet regions are transferred to the newly created component through the organizing panel and property (same as for lug-bracket) is designed to the component.
- The fillet regions are separated as identical components as in Fig. 14.
- Now the optimization panel is used to setup the optimization.
- Free shape optimization is selected from the optimization panel.
- The grid points in outer face of the new component as shown in Fig. 14 are selected as the deign space.
- The response of static stress is selected from the response panel.
- The objective of the optimization is stated as to minimize the maximum von-Mises stress.
- The file is saved and Optistruct solver is selected for optimization.

The shape optimization problem is solved and the result is obtained. The result in the shape change for the fillet areas as shown in Fig. (15-16).

The objective of the problem, namely minimizing the maximum von-Mises stress has been achieved.

The maximum von-Mises stress for the lug bracket after shape optimization is achieved as 118.151 MPa as shown in Fig. 17.

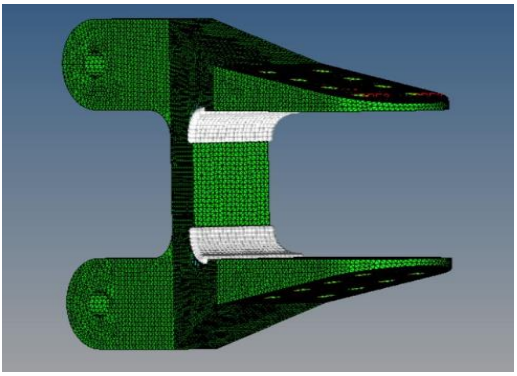

Fig. 13 – Design space

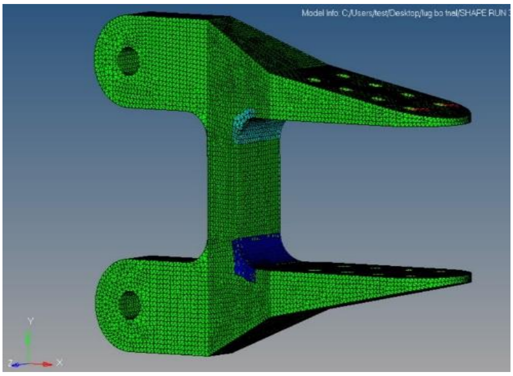

Fig. 14 – Divided into 3 individual components

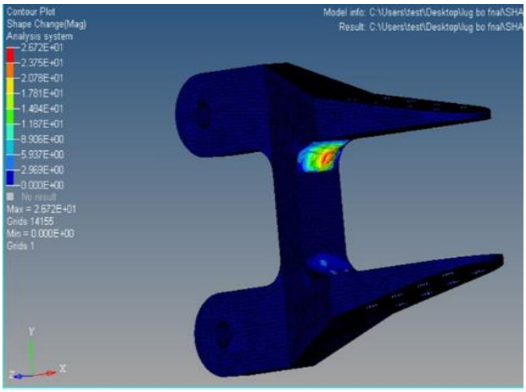

Fig. 15 – Shape change in upper fillet

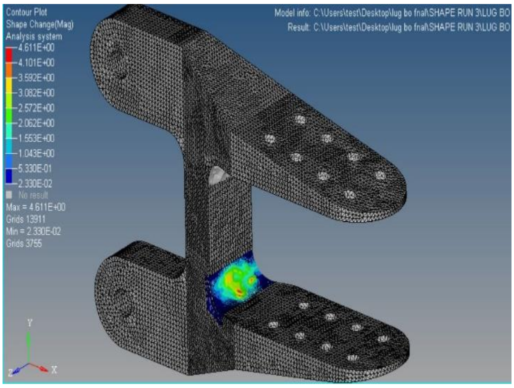

Fig. 16 – Shape change in bottom fillet

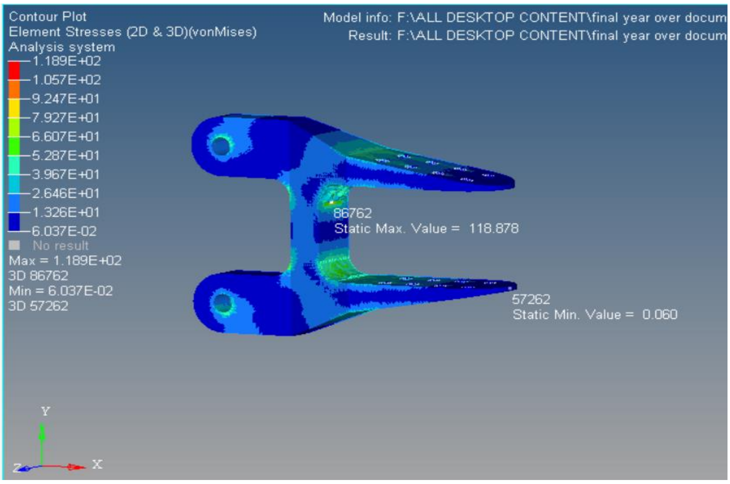

Fig. 17 – Minimized max von-Mises stress

12. RESULTS

In the previous section we saw the results of the finite element analysis and optimization of the models for the lug bracket.

In this section we will compare the results of the models for the lug bracket and validate a final design for the lug bracket. The comparison of the results of the lug bracket models is tabulated in Table 6.

Table 6. – Comparison of results

S. No	Models	Weight (kg)	Volume (mm^3)	Von-Mises stress (MPa)	Von-Mises strain	Displacement (mm)	Shear stress (MPa)
1.	Recreated	0.851	1.15	183.791	0.02	4.358	95.022
2.	Optimized	0.651	0.908	167.45	0.02	1.685	84.499

The most important factor for evaluating the service life of the lug bracket design can be defined as the load-bearing capacity of the component or the intensity of the component to withstand the stress.

From the tabular column we can see that the volume and von-Mises stress for the optimized lug bracket is very low compared to the other two models.

The stress concentration of the optimized model of the lug bracket is seen in the fillet regions; thus the fillets of the lug bracket are also optimized using the free size optimization by increasing the inner radius of the fillet, which is shown in the previous section. The optimized model for the lug bracket has a minimized compliance factor which states that the model has a higher stiffness factor.

The volume of the optimized model is compared with the other two models and it is found that it has a minimum volume of 0.908mm^3. The von-Mises stress and a shear stress value for the optimized model are constrained to 167.45 MPa and 84.499 MPa, respectively which states that the optimized model has a higher capacity to load for a longer period of time than the initial model. The displacement value of the optimized model is achieved as 1.685 mm which states that the model is highly rigid and has a high tensile strength.

13. CONCLUSIONS

The Lug-Bracket assembly is used to establish contact between the wing and fuselage of the aircraft and this is why it is said to be the most critical and complex load- bearing structure, has a high degree of safety factor and is manufactured with intense care and it also has a a higher resilience. From this research paper we can conclude that the optimization technique can be used to reduce the weight of the Lug bracket assembly and to minimize the maximum stress acting on the fillet regions of the Lug bracket. The optimized lug bracket model has a low compliance factor which means the weight of the bracket is reduced but the stiffness of the bracket is not compromised , which means the lug bracket assembly has a higher strength to weight ratio and higher load bearing capacity. Thus, the optimized model of lug bracket is highly efficient and can withstand longer working hours in heavy loading conditions; also, the light weight of the bracket makes it suitable for easy and fast manufacturing.

REFERENCES

[1] A. Chinnamahammad Bhasha, K. Balamurugan, Fracture Analysis of Fuselage wing joint developed by aerodynamic structural materials, *Materials Today: Proceedings*, 1 September 2020.

[2] J. Zhu, H. Zhou, C. Wang, L. Zhou, S. Yuan, W. Zhang, A Review of topology optimization for additive manufacturing : Status and Challenges, *Chinese Journal of Aeronautics*, 13 October 2020.

[3] L. Song, T. Gao, L. Tang, X. Du, J. Zhu, Y. Lin, G. Shi, H. Liu, G. Zhou, W. Zhang, An all-movable rudder designed by thermo-elastic topology optimization and manufactured by additive manufacturing, Computers & Structures, Volume **243**, 24 October 2020.

[4] C. Wang, X. Qian, Simultaneous optimization of build orientation and topology for additive manufacturing, *Additive Manufacturing*, Volume **34**, 11 May 2020.

[5] M. Nirish, R. Rajendra, Suitability of metal additive manufacturing processes for part topology optimization – A comparative study, Materials Today:Proceedings, Volume **27**, Part 2, Pages 1601-1607, 12 April 2020.

[6] C. M. Sample, V. K. Champagne, A. T. Nardi, D. A. Lados, Factors governing static properties and fatigue, fatigue crack growth and fracture mechanisms in cold spray alloys and coatings/repairs: *A review, Additive Manufacturing*, Volume **36**, 12 June 2020.

[7] Z. Wang, A. S. J. Suiker, H. Hofmeyer, T. Hooff, B. Blocken, Coupled aerostructural shape and topology optimization of horizontal-axis wind turbine rotor blades, Energy Conversion and Management, Volume **212**, 15 May 2020.

[8] S. Singamneni, L. V. Yifan, A. Hewitt, R. Chalk, W. Thomas, D. Jordison, Additive Manufacturing for the Aircrfat Industry: A Review, *Journal of Aeronautics & Aerospace Engineering*, Volume **8**, Issue 1, 18 February 2019.

[9] S. M. Fijul Kabir, K. Mathur, A.-F. M. Seyam, A critical review on 3D printed continuous fiber-reinforced composites: History, mechanism, materials and properties, *Composite Structures*, Volume **232**, 19 September 2019.

[10] N. Shyamsunder, R. Suresh, U. Bhaskar, Design and Analysis of Landing Gear Lug Attachment Bracket for Small Transport Aircraft, *International Journal of Mechanical Engineering and Technology (IJMET)*, Volume **9**, Issue 3, March 2018.

[12] M. Khan, M. Rehman Khan, D. Smitha, Design and Analysis on Aircraft Wing to Fuselage Lug Attachment, *IOSR Journal of Engineering*, Volume **1**, 4th International Conference Emerging Trends in Mechanical Science (ICEMS-2018).

[13] K. V. Prashant Reddy, I. M. Mirzana, A. Koti Reddy, Application of Additive Manufacturing technology to an Aerospace component for better trade-off's, *Materials Today:Proceedings*, Volume **5**, Issue 2, Part 1, Pages 3895-3902, 24 March 2018.

[14] C. Oberg, T. Shams, N.ader Asnafi, Additive Manufacturing and Business Models:Current Knowledge and Missing Perspectives, *Technology Innovation Management Review*, June 2018.

[15] Y.-C. Wang, T. Chen, Y.-L.an Yeh, Advanced 3D printing technologies for the aircraft industry: A fuzzy systematic approach for assessing the critical factors, *The International Journal of Advanced Manufacturing Technology*, Volume **105**, 05 April 2018.

[16] C. Shashikumar, N. Nagesh, Ganesh, Design and Analysis of Wing fuselage attachment bracket for fighter aircraft, *IJERGS*, Volume **4**, Issue 1, 2016.

[17] B. K. Venkatesha, K. P. Prashanth and T. Deepak Kumar, Investigation of Fatigue Crack Growth rate in Fuselage of Large Transport Aircraft using FEA Approach, *Global Journal of Researches in Engineering*, Volume **14**, Issue 1, Online ISSN:2249-4596 & Print ISSN:0975-5861, 2014.

[18] F. Hurlimann, R. Kelm, M. Dugas, G. Kress, Investigation of local load introduction methods in aircraft pre-design, *Aerospace Science and Technology*, Volume **21**, Issue 1, Pages 31-40, September 2012.

[19] S. K. Bhaumik, M. Sujata, M. A. Venkataswamy, Fatigue failure of aircraft components, *Engineering Failure Analysis*, Volume **15**, Issue 6, Pages 675-694, September 2008.

[20] O. Gencoz, U. G. Goranson and R. R. Merrill, Application of finite element analysis techniques for predicting crack propagation in lugs, *International Journal of Fatigue*, Volume **2**, Issue 3, Pages 121-129, Boeing Commercial Airplane Company, Seattle, Washington, 98124, USA, July 1980.