ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering

Volume 5 – No.3, May 2018

BUCKLING ANALYSIS AND LIFE ESTIMATION OF ATTACHMENT BRACKET USED IN AIRCRAFT STRUCTURE

B.Anusha
PG Scholar
Methodist College of Engineering
and Technology, Hyderabad.
anushabandari9@gmail.com

A.Rajasekhar
Professor, Department of
Mechanical Engineering
Methodist College of Engineering
and Technology, Hyderabad.
arsekhar06@gmail.com

ABSTRACT

Attachment brackets are one of the most fracture critical components in aircraft structure, and the failure of attachment bracket can be very severe that sometimes the wing and fuselage get separated and result in accidents. Finite element analysis helps the designer to safeguard the structure from lamentable failure. Therefore, it is important to establish design criteria and analysis methods for life estimation of the bracket. In current study, the attachment bracket which is used to connect wing and fuselage of an aircraft is considered and 3-dimensional model of the bracket is designed using NX-CAD. Finite element analysis of the attachment bracket under the typical loading condition was carried out using Ansys for two different materials (Steel Alloy AISI-4340 and Aluminum Alloy-2024-T351). This paper mainly focuses on optimizing the life of the attachment bracket by performing buckling and fatigue analysis. From the analysis, it is observed that the life of Steel Alloy is found to be more when compared to Aluminum alloy.

Keywords

Attachment bracket, Finite element analysis, Buckling Factor, Fatigue analysis, Life estimation.

1. INTRODUCTION

Attachment brackets are the primary structural elements in airframe structure that are widely used in connecting different components of the airframe. Failure of attachment bracket may lead to catastrophic failure of the aircraft. Stress Analysis is performed on Wing Attachment Bracket and found that maximum stress is acting at circular hole of Spare plate [1]. By static Structural and Modal Analysis of Engine Bracket using Finite Element Analysis it is found that low natural frequency will prove as a hindrance in vibration characteristics of the bracket [2].The consequences of structural bracket failure can be very severe that sometimes the wing and fuselage get separated and result in accidents. Therefore, it is important to establish design criteria and analysis methods to ensure the damage tolerance of aircraft lug attachment. In current study, an attachment bracket is designed against fatigue failure in 3-D model using NX-CAD. Static analysis is done by importing 3-D model in Ansys. Buckling and Fatigue analysis are done for two most commonly used materials (Steel alloy AISI 4340 and Aluminum alloy-2024-T351) [5] for the life prediction and the best material for attachment bracket is found.

2. DESIGN

The model of the attachment bracket is designed in NX-CAD. The data obtained from the different references is found to be similar, so here a new approach was made and modeled the component by considering the data from united launch alliance design challenge on attachment brackets. United Launch Alliance is a space Launch Company by Boeing with more than 120 consecutive launches. The 2D input and the isometric view of the bracket are shown in Figure1 and Figure2 respectively.

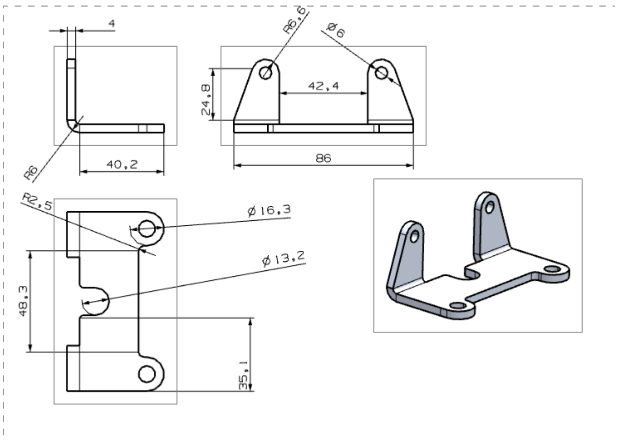

Fig 1: 2D input for the bracket

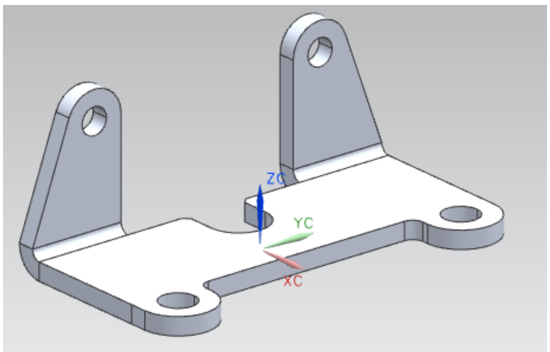

Fig 2: Isometric view of Attachment Bracket

ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering

Volume 5 – No.3, May 2018

3. FINITE ELEMENT ANALYSIS

Two Different materials (Steel Alloy AISI 4340 and Aluminum alloy-2024-T351) are chosen for the bracket and Finite Element Analysis is performed on it for 6 different load cases. The load calculations are performed for finding the load acting on the attachment bracket by considering a medium aircraft of 37278 N [1] and Factor of safety as 1.5, the load values for 6g cases are shown in table1.

Table1: Load values for different load cases of an Attachment bracket.

S.No.	Parameter (g)	Load Value (N)
1	1g	4543.21 N
2	2g	9086.43 N
3	3g	13629.63 N
4	4g	18172.84 N
5	5g	22716 N
6	6g	27259.3 N

Now, the deflections obtained in steel and Aluminum alloy for 6g load case are shown in figures 3 and 4 respectively.

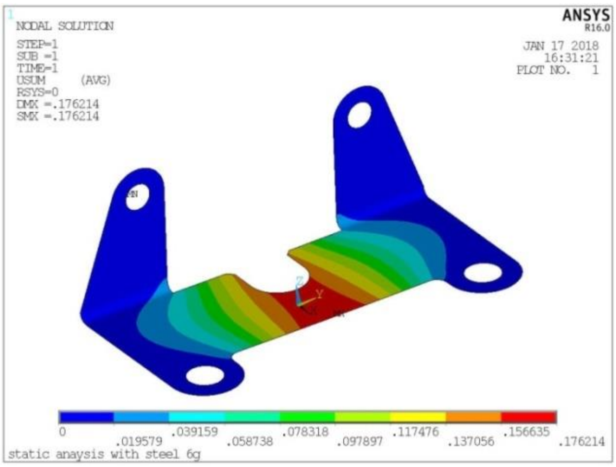

Fig 3: 6g load U_{sum} deflection in Steel alloy

The deflections obtained in steel and Aluminum alloy are shown in figure5.

The Von mises stresses developed in Steel and aluminum alloy are obtained and are tabulated in table2.

From static analysis, it is found that for AISI-4340 material under the worst load condition i.e. for 6g, the maximum deflection obtained is 0.176 mm and the von Mises stress developed is 818.155 MPa. For 2024-T351 material, the maximum deflection obtained is 0.490 mm and the von Mises stress developed is 819.014 MPa.

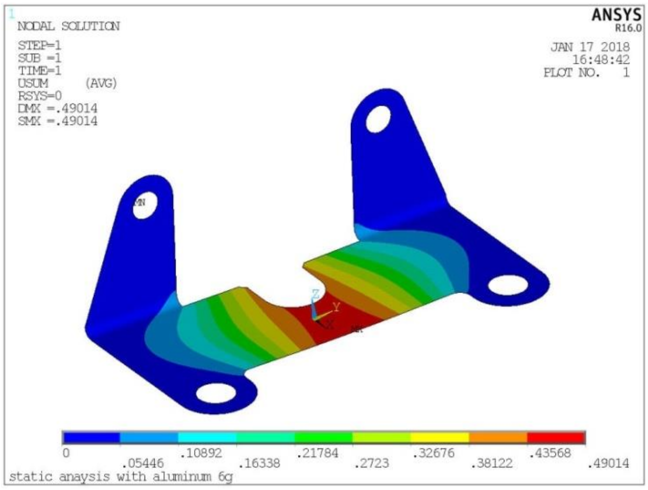

Fig 4: 6g load U_{sum} deflection in Al alloy

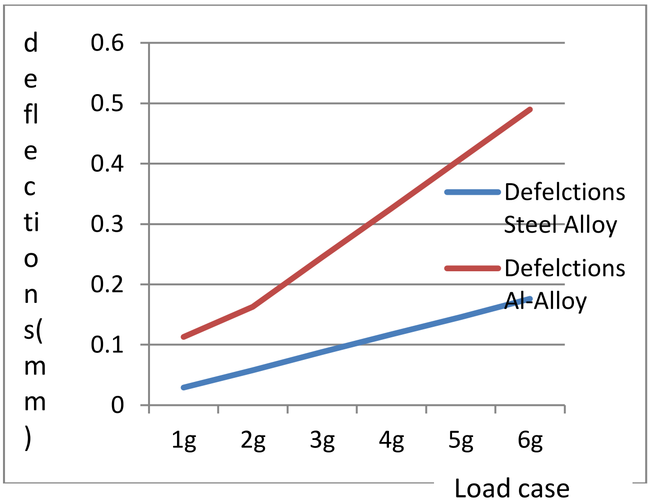

Fig 5: Deflections in Steel and Al Alloy

Table2: Von-mises stresses in Steel and Al-Alloy

Load Case	Von-mises stresses	
	Steel Alloy	Al-Alloy
1g	223.6	136.4
2g	273	272.7
3g	409.5	409.1
4g	546	545.4
5g	682.5	681.8
6g	819	818.2

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 5 – No.3, May 2018

4. BUCKLING ANALYSIS

Buckling analysis is done for the two materials without changing the boundary conditions applied in the static analysis. Buckling load factor (BLF) is determined for the both materials for different load conditions. Buckling Load Factor (BLF) is the factor of safety against buckling or the ratio of the buckling loads to the applied loads. So, if the BLF is high it indicates that the safety rate is higher.

The buckling condition for load case 6g for the steel and Aluminum alloy is depicted in figures 6 and 7.

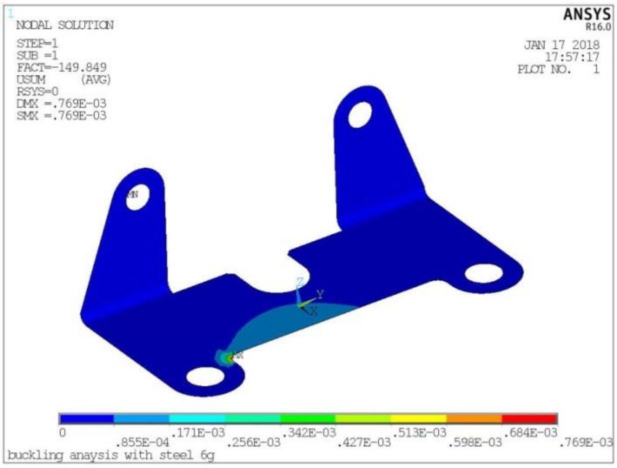

Fig 6: Buckling load condition of 6g-steel alloy

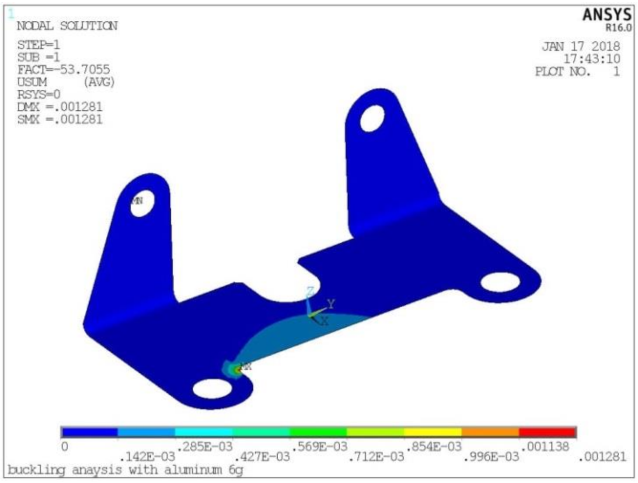

Fig 7: Buckling load condition of 6g-Al alloy

The buckling load factors for Steel and Aluminum alloy for different load cases are shown in figure8.

From the buckling analysis, it is found that the structure is buckling at BLF of 149.84 at a load condition of 6g for AISI-4340. For 2024-T351, the structure is buckling at BLF of 53.7 at a load condition of 6g.

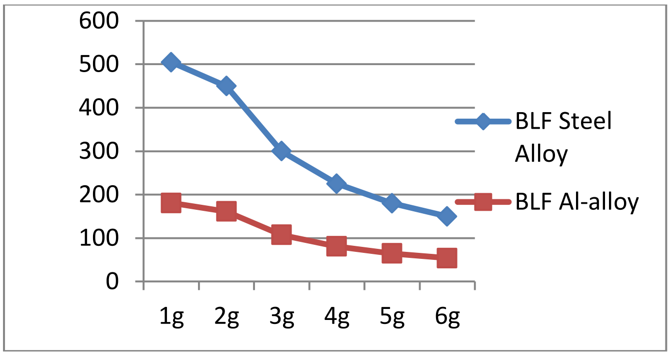

Fig 8: Comparison of BLF for Steel and Al-alloy

5. FATIGUE ANALYSIS

In Ansys, fatigue analysis is performed as a post processor operation considering the stress cycle developed from different load conditions. In order to vary the stress levels and create stress cycles the following conditions are considered:

- The first event having the no load condition
- The second event having the load of 6g

The maximum number of cycles allowed for load case 6g for steel and Aluminum alloy are shown in figure 9 and figure 10 respectively.

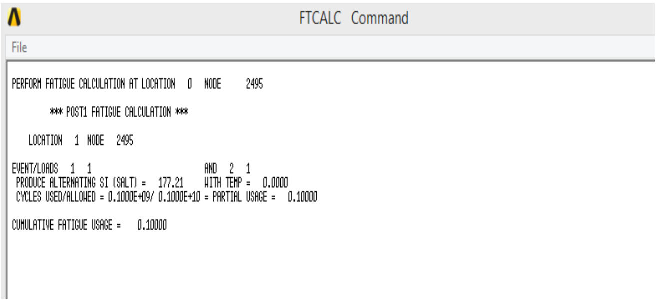

Fig 9: Fatigue Analysis results for steel Alloy

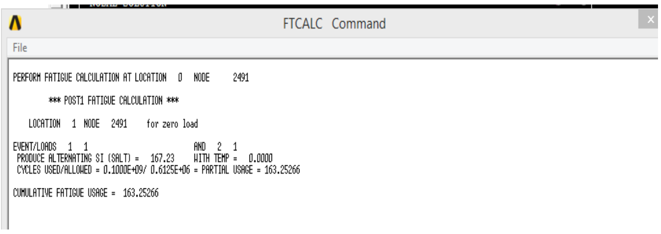

Fig 10: Fatigue Analysis results for Al Alloy

From the fatigue analysis, it is found that the maximum numbers of cycles allowed for AISI-4340 at the load condition of 6g are $0.1e^{10}$ cycles and the maximum numbers of cycles allowed for 2024-T351 material at the load condition of 6g are 0.6125^{e6} cycles.

6. CONCLUSION

From the results, it is found that Von Mises stress developed in both the materials is approximately same i.e. 818.155 MPa(for AISI 4340) and 819.014 MPa(for 2024 T351), but the deflection developed in the 2024 T351 material (0.490mm) is more than that of AISI 4340 (0.176mm). AISI 4340 is having high buckling factor than that of the 2024 T351 material. It is clearly found that the life of the steel bracket is more compared to the aluminum alloy. Thus, it can be concluded that the AISI steel 4340 is the first choice for the attachment bracket than the Aluminum 2024 T351, provided the weight of the bracket is in acceptable limits. The work can be extended for different compositions of aluminum alloy to optimize the weight of the bracket.

7. REFERENCES

[1] Harish E.R.M, Mahesha. K, Sartaj Patel. "Stress Analysis for Wing Attachment Bracket of a six seater Transport Airframe Structure". (IJERT) Vol. 2, Issue 7, July 2013 ISSN: 2319-8753.

[2] B.K. Sriranga, Dr.C.N. Chandrappa, R. Kumar and Dr.P.K. Dash. “Stress Analysis of Wing-Fuselage Lug Attachment Bracket of a Transport Aircraft". (ICCOMIM - 2012), 11-13 July, 2012.

[3] Umesh S. Ghorpade, Prof. D.S. Chavan, Prof. M.V. Kavade, “Static Structural and Modal Analysis of Engine Bracket Using Finite Element Analysis”

[4] Amardeep singh, Jashandeep singh and Paviterpuneet singh have published a paper on "Lug Attachment of Wing-Fuselage".International Journal of Aerospace and Mechanical Engineering Volume 1 – No.1, September 2014ISSN (O): 2393-8609.

[5] ShashiKumar C., Nagesh N., “Design and Analysis of Wing Fuselage attachment bracket for fighter aircraft” Kuppam Engineering College, Kuppam.

[5] J. Vogwell, J. M. Minguez , "Failure in lug joints and plates with holes". School of Mechanical Engineering, University of Bath, Bath BA2 7AY, U.K., Facultad de Ciencias, Universidad Del Pais Vasco, Bilbao, Spain.

[6] C. R. F. Azevedo, E. Hippert, Jr., G. Spera and P. Gerardi,"Aircraft landing gear failure: fracture of the outer cylinder lug. Laboratory for Failure Analysis", Instituto de Pesquisas Tecnológicas, PO Box 0141, São Paulo, Brazil.