International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 2, No. 2, April 2013

Research Paper

ANALYSIS OF THE WINGBOX WITH SPLICED SKIN AND ESTIMATION OF THE FATIGUE LIFE FOR THE WINGBOX

S Sarath[1]*, Jason Cherian Issac[1] and K E Girish[2]

Corresponding Author: **S Sarath, ✉ sarathmex@yahoo.com*

The attachment joints are inevitable in any large structure like an airframe. Splicing is normally used to retain a clean aerodynamic surface of the wing skin. The wings are the most important lift-producing part of the aircraft. Wings vary in design depending upon the aircraft type and its purpose. The wing box has two crucial joints, the skin splice joint and spar splice joint. Top and bottom skins of inboard and outboard portions are joined together by means of skin splicing. Front and rear spars of inboard and outboard are joined together by means of spar splicing. The skins resist much of the bending moment in the wing and the spars resist the shear force. In this study the chord-wise splicing of wing skin is considered for a detailed analysis. The splicing is considered as a multi row riveted joint under the action of tensile in plane load due to wing bending. Stress analysis of the joint is carried out to compute the stresses at rivet holes due to by-pass load and bearing load. The stresses are estimated using the finite element approach with the help of PATRAN/NASTRAN. In a structure like airframe, a fatigue crack will appear at the location of high tensile stress. Further these locations are invariably the sites of high stress concentration. Life prediction requires a model for fatigue damage accumulation, constant amplitude S-N (stress life) data for various stress ratios and local stress history at the stress concentration The response of the splice joint will be evaluated. The splice joint is one of the critical locations for fatigue crack to initiate. In this study prediction of fatigue life for crack initiation will be carried out at maximum stress location.

Keywords: Splice joint, Finite element analysis, Wingbox, Fatigue, Stress analysis, Life prediction

INTRODUCTION

The wingbox is a structural component in an aircraft designed to provide support and rigidity to the wings. Designs vary, depending on the size and function of an aircraft, but generally this is the strongest section of the

[1] SAINTGITS College of Engineering, Kottayam 686002, Kerala, India.
[2] Banglore Aircraft Industries, Ltd. (P), Bangalore, India.

fuselage and it can include a number of supportive spars, as well as chambers designed to isolate impacts. Usually, this component is not readily visible, although we can assume it lies between the wing roots, the parts of the plane where the wings attach. Aircraft in flight experience concentrated shear stresses on their wings. Without adequate support, the wings would eventually fold up against the side of the plane. The wingbox absorbs some of this stress and distributes it across a supportive framework, preventing the wings from wobbling or bending. In addition to holding the wings in place, it helps absorb impacts sustained during events like turbulence to keep the plane in the air. In a wingbox, most of cases the stringers are attached to the skin through rivets. These joints will help in to transmit forces mainly along there length. Forces parallel to the skin and directed at right angles to stringers will be limited by torsional flexibilty of these members. Forces normal to the skin will be limited in magnitude by the small bending strength of the skin and stringers. Splicing is normally used to retain a clean aerodynamic surface of the wing skin. The splicing is considered as a multi row riveted joint under the action of tensile in plane load due to wing bending. They are prone to crack due to fatigue (Michael, 1988).

Fatigue is a phenomenon associated with variable loading or more precisely to cyclic stressing or straining of a material. Just as we human beings get fatigue when a specific task is repeatedly performed, in a similar manner metallic components subjected to variable loading get fatigue, which leads to their premature failure under specific conditions. Fatigue cracks are most frequently initiated at sections in a structural member where changes in geometry, e.g., holes, notches or sudden changes in section, cause stress concentration (Jaap, 2004).

LITERATURE REVIEW

Adarsh *et al.* (2012) has studied about the stress analysis and fatigue life prediction for splice joint in an aircraft fuselage through an FEM approach. Aluminium alloy 2024-T351 material is considered for all the structural elements of the panel for fabrication of the aircraft body. Force due to cabin pressurization is considered as one of the critical load cases for the fuselage structure. Fuselage experiences constant amplitude load cycles due to pressurization. Splice joints are used for the fuselage structure. Typical splice joint panel consisting of skin plates, doubler plate and a longitudinal stiffener is considered for the study. The project includes the stress analysis of a splice joint in a transport aircraft. A two-dimensional finite element-analysis is carried out on the splice joint panel. Distribution of fasteners loads and local stress field at rivet locations are studied using finite element analysis. The work involved the analysis of the splice joint using software's MSC/NASTRAN and MSC/PATRAN. A two-dimensional finite element-analysis is carried out on the splice joint panel. Distribution of fasteners loads and local stress field at rivet locations is studied from finite element analysis. The work also involves the modifications required to correct the boundary effects of the panel. The global finite element analysis of a segment of typical fuselage will be carried out. This global finite element analysis results will be bench mark for comparing the results from the splice joint panel analysis. Repeated finite element analysis is carried out to get the response of the parent structure (fuselage) at the joint location. The response of the splice joint is evaluated. The splice joint is one of the critical locations for fatigue crack to initiate. Hence prediction of fatigue life for crack initiation is carried out at maximum stress location.

New concepts were proposed related to structural design, materials, production techniques, inspection procedures and load spectra by Jaap (2009). The paper presents a personal impression of evaluating experience, design aspects, predictions and experiments. It explains that predictions of fatigue properties and experimental verifications are most important tools.

Xiong and Bedair (1999) mentioned about modeling procedures for the stress analysis of riveted lap joints in aircraft structure. Analytical methods have been developed based on the complex variational approach for lap joints with single or multiple rivet holes. The joined plates can be either metallic or composite materials. The stresses in the two joined plates and the rivet loads is determined. Finite element analyses are conducted using the commercial packages MSC/Patran and MSC/Nastran.

Amarendra (2006) conducted a study where the main objective of the research was to establish a link between critical riveting process parameters on the potential of fatigue damage in the joint. Aircraft fuselage splices are fatigue critical structures and the damage associated with these structures has been widely recognized as a safety issue that needs to be addressed in the structural integrity of aging aircraft. An effective means for structural evaluations of airworthiness of aging aircraft and obtaining essential data for evaluation of such type of fatigue cracking is airframe teardown inspections and laboratory fatigue testing of lap joint. The Federal Aviation Administration and Delta Airlines teamed up in such an effort to conduct destructive evaluation, inspection and extended fatigue testing of a retired Boeing 727-232 (B727) passenger aircraft near its design service goal. Preliminary visual inspection revealed a large number of cracks in the aircraft fuselage lap joint emanating from the rivet/skin interface. Most of these cracks were observed in the lower skin such that they could not be detected under an operator's routine maintenance. The presence of these cracks was attributed to the sharp gradients of stress arising from contact between the installed rivet and rivet holes. The residual stress field generated during the rivet installation has a strong impact on the nucleation and propagation of fatigue cracks at and around the rivet/skin interface. Floyd (2010) determined the applicability of an approach to predict the number of cycles of fatigue loading of a structure to failure.

PROBLEM DEFINITION

In this study the chord-wise splicing of wing skin is considered for a detailed analysis. The splicing is considered as a multi row riveted joint under the action of tensile in plane load due to wing bending. Stress analysis of the joint is carried out to compute the stresses at rivet holes due to by-pass load and bearing load. The main objective are:

- Global and local stress analysis of the splice joint in an aircraft wingbox to compute the stresses at rivet holes due to tension with the help of MSC PATRAN and MSC NASTRAN.

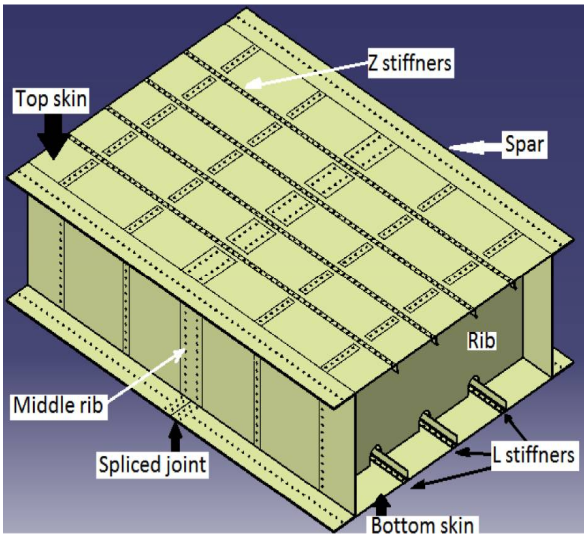
- Fatigue life prediction for crack initiation at the splice joint region by Miner's Rule.

Al 2024-T351 is used in current wingbox due to high strength and fatigue resistance properties. The ultimate tensile strength of this material is 68 ksi (470 MPa) and yield strength is 41 ksi (280 MPa) and it has an elongation of 19% (Michael, 1993).

Geometrical Configuration

Wingbox modeled in CATIA was been shown in Figure 1. It consists of different structures. Wingbox used here consists of five ribs including a middle rib, stiffners, bottom and top skins, spars. Each part is modeled in CATIA software and assembled to form wingbox.

Figure 1: Wingbox Modeled in CATIA V5



Loads on the Wingbox

Lift load is considered as important criteria while designing an aircraft. Fuselage and wings are the two main regions where lift load acting in an aircraft. Here 80% of the lift load is acted on the wings (i.e., maximum lift load is acted on the wings) and remaining 20% in acted on the fuselage. Therefore in wings the maximum load is acted nearer to the wing roots and minimum load is acted at the tip of a wing box.

Load calculation for the wingbox

Weight of the aircraft: 19620 N

Design load factor: 3"g"

Factor of safety: 1.5

Therefore,

Total design load on the aircraft will be: 88290 N

As we mentioned earlier, total lift load on the aircraft is distributed as 80% and 20% on wing and fuselage respectively,

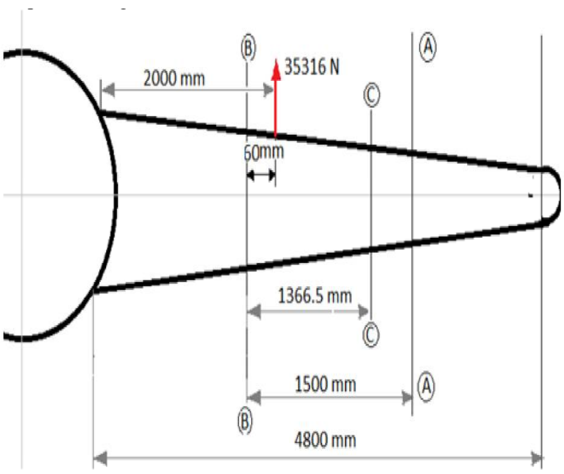
Hence total load acting on the wing = 70632 N

Therefore total load acting on the each wing = 35316 N

But we know the resultant load is acting at the distance 2000 mm from the wing root as shown in Figure 2.

Bending moment at the root of the wing can be calculated as 70.632×10^6 Nmm

Figure 2: Load Case for Current Wing



The load required at section C-C to stimulate the bending moment is P = 21361.57 N

Total length of C-C cross section of wingbox including all the parts = 6006 mm

Load distributed on the cross section = 3.5567 N

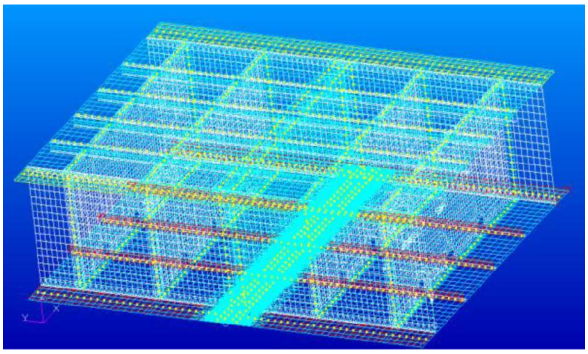
Finite Element Package

In this project MSC PATRAN software is used as the preprocessor and postprocessor. The preprocessing task includes building the geometric model by importing it from CATIA solid model of wingbox and extracting geometry, building the finite element model, giving these elements the correct properties, setting the boundary conditions and loading conditions and finally, assembling these elements into a connected structure for analysis. Analysis is done in MSC NASTRAN. The analysis stage simply solves for the unknown degrees of freedom, as well as reactions and stresses. In the post processing stage, the results are evaluated and displayed. The accuracy of these results is postulated during this post processing task. The PATRAN and NASTRAN software together perform all 3 of the principle tasks of a finite element analysis (MSC Software, 2010).

ANALYSIS OF WING BOX

The geometric model of the wingbox done in CATIA V5 software package is imported to MSC PATRAN for preprocessing. Each parts are extracted by its points and lines to get geometrical accuracy to the model. In our wingbox we do have five ribs including one middle rib. Other than middle rib all other four ribs are identical. Each rib consist of two flanges, two sides and a web. This has to be meshed separately creating different groups. Special care is to be taken for meshing the semicircular region in the web or the stiffner cut-out. Middle rib have more wider flange and ribs as it has to support the skin splice joint. Finite element properties are provided to two spars used here are C-section structures. While meshing top skin as well as bottom skin of the wingbox it is taken care that mesh seeds are provided for the all the riveting positions for the later simplicity. As we know we are using L-stiffners on bottom part and Z-stiffners on top part of the wingbox, these stiffners are spaced at a certain gap and running parallel to spar. Special cut out are provided in the ribs to provide a fine run along the skin. We have three L-stiffners and four Z-stiffners in our current wingbox. Rivets are made by using one dimensional beam element. We had already provided mesh seeds to create node for rivets wherever needed in all other parts. All the joints are riveted in wingbox, hence a large number of rivets are used. All the elements of the parts are verified for boundaries, normals, connectivity and duplicates. All the elemental and material properties (Aluminium 2024 T351) are provided for analysis. Figure 3

Figure 3: Assembled Wingbox with Finite Element Properties



shows the whole finite element mesh generation.

The stress distribution for the given loads have been observed and that reveals the stress is distributed uniformly but maximum stresses are developed nearer to spliced joint exactly at the rivets which connects spar and bottom skin as shown in Figure 4. The magnitude of maximum principal stress developed here is 193 N/mm^2. We had also conducted an analysis keeping all the rivet rotation constrained in rotating direction (x axis). Here also the maximum stress is developed on the same location and same rivet but the stress magnitude is decreased considerably to 145 N/mm^2. Since the maximum stress we getting is at the same location we do local analysis on the specific location. It includes skin spliced region which we can take as two sheet plates and the lower spar region which we can consider as a L section. Corresponding rivets holes are provided at the exact location. The geometry is as shown in Figure 5. The exact location of maximum principal stress from global analysis is created in PATRAN for local analysis with the help of geometrical tools. It is then meshed separately forming different groups. Final solid modeling including corresponding parts for local analysis has been shown in Figure 6.

Figure 4: Maximum Principal Stress Contour of Wingbox

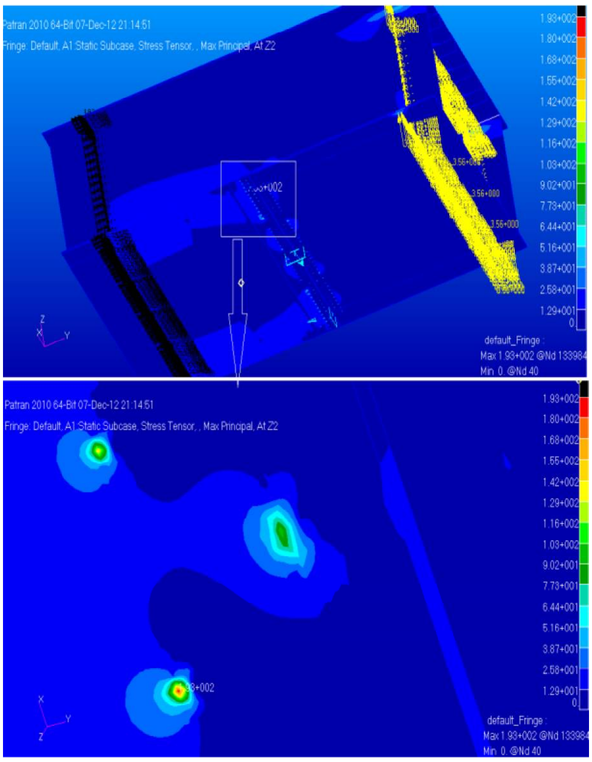


Figure 5: Geometry for Local Analysis

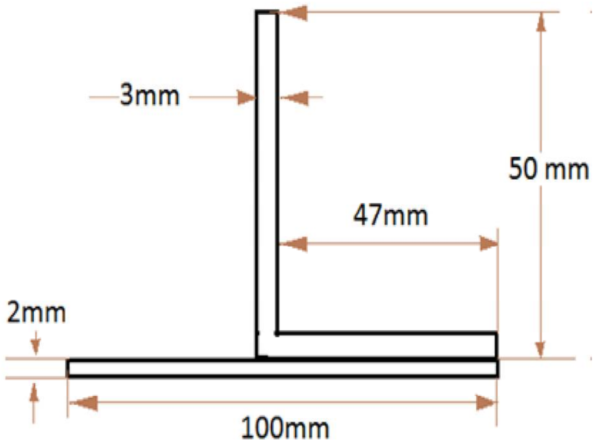
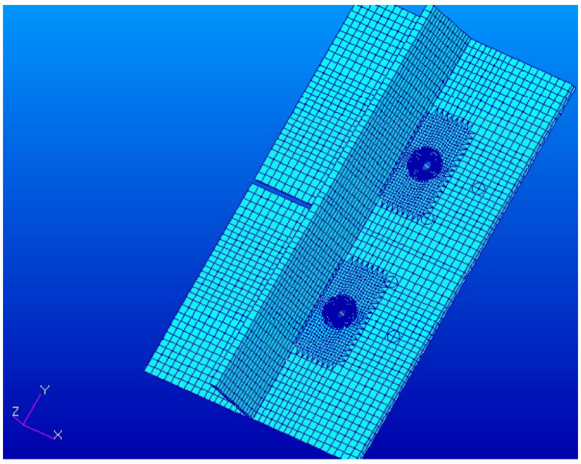


Figure 6: Whole Finite Element Model for Local Analysis



Average Stress value at a distance 85 mm to 90 mm from the spar tip is taken where maximum stress is identified in global analysis for calculation of load in local analysis and which is found to be 8.5 N/mm^2.

Hence total load = Stress $\times$ area = 1700 N

Distributed load on cross section for local analysis = 17 N

For the load case, one end of the section is completely constrained and the other end is applied with the calculated load of 17 N/mm from calculation in order to create tension at the other tip. Hence one end is constrained in all its six degree of freedom, i.e., translation and rotation.

As in the case of global analysis, the particular area considered for local analysis undergoes tension in bottom skin. In order to create the same surrounding, we constrain any one or two translation direction, hence we took two cases.

Case 1: With z translation constraint

Case 2: With x and z translation constraint

During different iteration the maximum principal stress is found out to be 503 N/mm^2

Figure 7: Maximum Principal Stress Contour for Case 1

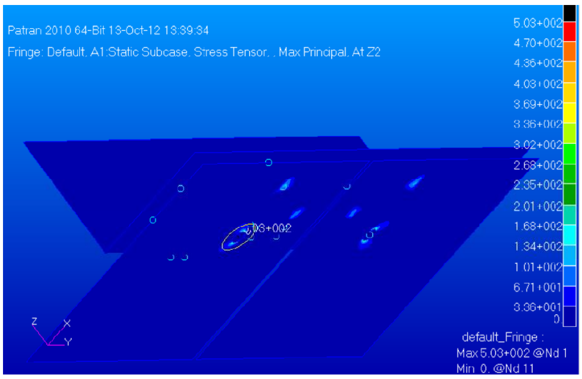
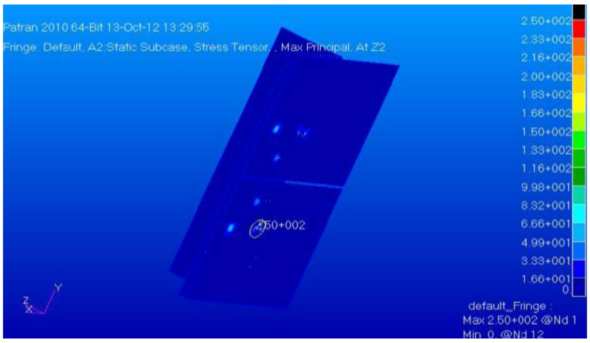


Figure 8: Maximum Principal Stress Contour for Case 2



in case 1 and 250 N/mm^2 in case 2 as shown in Figures 7 and 8 respectively.

The structure is safe because the stress magnitude which was obtained from the analysis is less than the yield strength of the structural material. Once wing box is safe from linear static analysis, next step is the fatigue life prediction of the wing box.

FATIGUE LIFE CALCULATIONS

From the global and local stress analysis of the wingbox the maximum tensile stress location is identified. A fatigue crack will always initiate from the location of maximum tensile stress. From the stress analysis it is found that such a location is at a rivet hole. Normally aircraft wing experiences variable spectrum loading during the flight. A typical transport aircraft flight load spectrum is considered for the fatigue analysis of the wingbox. Calculation of fatigue life is carried out by using Miner's Rule. For the fatigue calculation the variable spectrum loading is simplified as block loading. Each block consists of load cycles corresponding to 100 flights. Damage calculation is carried out for the complete service life of the aircraft. The load factor g, is defined as the ratio of the lift of an aircraft to its weight and represents a global measure of the load to which the structure of the aircraft is subjected.

The maximum principal stress in local analysis is 503 N/mm^2 is taken for the calculation, as this is the most safest value to predict the life. As we know the maximum stress value obtained from the analysis is corresponding to 4.5 g condition. Therefore the stress value corresponding to 0.5 g condition is obtained as 55.89 N/mm^2.

Correction factors for fatigue life calculations of wingbox is considered. Hence maximum stress with correction factor is calculated (Jaap, 2004).

- Surface roughness correction factor = 0.8
- Type of loading = 1
- Correction factor for reliability in design = 0.897

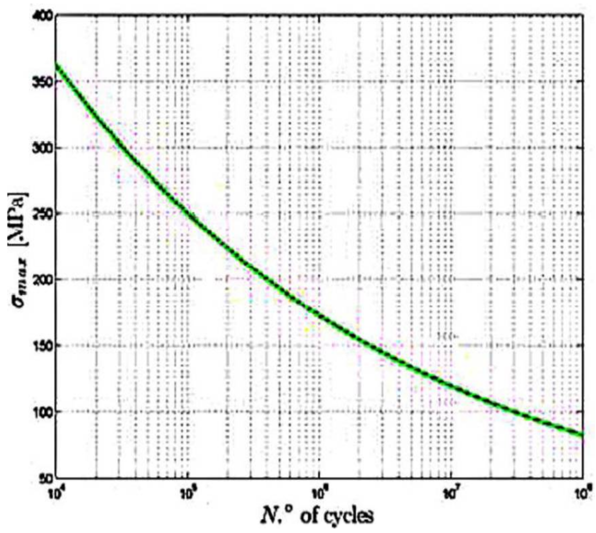
Maximum stress with correction factor for all the other conditions are calculated and shown in Table 1. When the alternating or maximum stress is plotted versus the number of cycles to failure (fatigue life) for a given material, the curve is known as S-N curve

Table 1: Stress Values at Various "g" Conditions with Correction Factors

"g" Condition	Maximum Stress (N/mm^2)	Maximum Stress with Correction Factor (N/mm^2)
0.50	55.889	77.88
0.75	83.834	116.83
1.00	111.778	155.77
1.25	139.722	194.70
1.50	167.667	233.65
1.75	195.611	272.60
2.00	223.556	311.50

(Michael, 1988). Using the maximum stresses value at different g conditions, corresponding number of cycles to failure is obtained from S-N curve of Aluminium 2024 T351 as shown in Figure 9 (Serrano *et al.*, 2010).

Figure 9: Typical S-N Diagram for Fatigue Behavior of Aluminium 2024 T351



The simplest and most practical technique for predicting fatigue performance is the Palmgren-Miner hypothesis. The hypothesis contends that fatigue damage incurred at a given stress level is proportional to the number of cycles applied at that stress level divided by the total number of cycles required to cause failure at the same level. If the repeated loads are continued at the same level unit failure occurs, the cycles ratio will be equal to one.

From Miner's equation (Jadav *et al.*, 2012),

$\Sigma n_i/N_f = C$

where, n_i = Applied number of cycles

N_f = number of cycles to failure

Table 2 shows damage D, accumulated on each range of load condition.

Table 2: Damage Accumulated from Miner's Formula

Range of "g"	Applied No. of Cycles (n_i)	No. of Cycles to Failure (N_i)	Damage Accumulated (n_i/N_i)
0.5 "g" to 0.75 "g"	48000	3×10^7	1.6×10^{-4}
0.75 "g" to 1.0 "g"	33000	3×10^6	0.011
1.0 "g" to 1.25 "g"	26000	6×10^5	0.0433
1.25 "g" to 1.50 "g"	22000	2×10^5	0.1
1.75 "g"	45	6×10^4	7.5×10^{-4}
2 "g"	1	4×10^4	2.5×10^{-5}
-0.5 "g" to 1.5 "g"	100	2×10^5	5×10^{-4}

Total damage accumulated for all load case is given by $D_a = D_1 + D_2 + D_3 + D_4 + D_5 + D_6 + D_7 = 0.155735$

Total damage accumulated is 0.155735, which is less than 1.

Therefore a crack will not get initiated from the location of maximum stress in the wingbox for given load spectrum.

Hence total damage is 0.155735 for 1 block of loading or for 100 flights.

One flight is considered 10 flying hours which eventually means 100 flights as 1000 flying hours.

For damage to become critical ($D = 1$), the number of blocks required is 6.421 blocks or 6421 hours.

Hence it is advised to meet the wingbox part maintenance atleast by this required time.

CONCLUSION

Stress analysis of the wingbox is carried out and maximum tensile stress is identified at one of the rivet holes near splice joints which is found out to be lower than yield strength of the material. Local analysis is conducted for the specific region for maximum principle stress. By local analysis it is validated that the maximum stress is at the same rivet hole during global analysis. Maximum tensile stress of 503 N/mm^2 is observed in the wingbox. Several iterations are carried out to obtain a mesh independent value for the maximum stress. A fatigue crack normally initiates from the location maximum tensile stress in the structure. The fatigue calculation is carried out for an estimation of life to crack initiation. Since the damage accumulated is less than the critical damage the location in the wingbox is safe from fatigue considerations. Life of the particular region in wingbox is predicted to become critical and found out to be 6421 flying hours or 6.421 blocks, hence advised to conduct the maintenance without fail during this period.

Fatigue crack growth analysis can be carried out in the other parts of the wingbox as future work. As well as damage tolerance evaluation and structural testing of the wingboxcan also be carried out for the complete validation of all theoretical calculations.

ACKNOWLEDGMENT

I express my sincere gratitude to all the engineers in BAIL especially to Mr K E Girish and Dr P K Dash for their support and guidance to carrying out the study in a successful manner. I stand grateful to my guide Prof Dr Jason Cherian Issac, for his guidance, motivation and valuable advice that he had given me throughout the work.

REFERENCES

1. Adarsh Adeppa, Patil M S and Girish K E (2012), "Stress Analysis and Fatigue Life Prediction for Splice Joint in an Aircraft Fuselage Through an FEM Approach", *International Journal of Engineering and Innovative Technology (IJEIT)*, Vol. 1, No. 4, pp. 142-144.

2. Amarendra Atre (2006), "A Finite Element and Experimental Investigation on the Fatigue of Riveted Lap Joints in Aircraft Applications", May, Dissertation, Georgia Institute of Technology.

3. Floyd Caiado (2010), "Fatigue Life Estimation of Notched Aluminum Sheet Specimens Subjected to Periodic Tensile Overloads", Proceedings of the 6th Annual GRASP Symposium, pp. 17-21, Wichita State University.

4. Jaap Schijve (2004), "Fatigue of Structures and Materials", Kluwer Academic Publishers.

5. Jaap Schijve (2009), "Fatigue Damage in Aircraft Structures Not Wanted but Tolerated", *International Journal of Fatigue*, Vol. 31, No. 6, pp. 998-1011.

6. Jadav Chetan S, Panchal Khushbu C and Patel Fajalhusen (2012), "Review of the Fatigue Analysis of an Automobile Frames", *International Journal of Advanced Computer Research*, Vol. 2, No. 4, pp. 103-107.

7. Michael Bauccio (1993), *ASM Metals Reference Book*, 3rd Edition, ASM International, Materials Park, OH.

8. Michael C Y Niu (1988), "Airframe Structural Design", Lockheed International System Company, Conmilit Press Ltd.

9. MSC Software (2010), "Company Overview Brochure", Expanding the Horizon of Engineering Simulation, MSC Press Release.

10. Serrano A S, Infante V I M N and Marado B S D (2010), "Fatigue Life Time Prediction of PoAF Epsilon TB-30 Aircraft – Part I: Implementation of Different Cycle Counting Methods to Predict the Accumulated Damage", CIFIE 2010, Iberian Conference on Fracture and Structural Integrity, March 17-19, FEUP, Porto, Portugal.

11. Xiong Y and Bedair O K (1999), "Analytical and Finite Element Modeling of Riveted Lap Joints in Aircraft Structure", *The American Institute of Aeronautics and Astronautics (AIAA) Journal*, Vol. 37, No. 1, pp. 93-99.