Stress Analysis of Fuselage Segment with Wing Attachment Bracket

[1]Praveenkumara B M, [2]Bhanuprakash M J, [3]Rajesh Kumbara S K, [4]Dr. B Sadashive Gowda
[1,2,3,4] Department of Mechanical Engineering, Vidyavardhaka college of Engineering Mysuru, Karnataka,India
[1]praveenkumarabm@vvce.ac.in, [2] bhanuprakash.mj@vvce.ac.in, [3]rajeshkumbara.sk@vvce.ac.in,

Abstract—The main objectives of this project are to present an identification of critical locations in ensuring the safety of aircraft structure and stress analysis is also carried out on structure. Finite element analysis is overall used for stress analysis of the structure and to find out fatigue critical locations. Due to fluctuating loads, fatigue cracks are main problems which occurred during service life of aircraft. Generally, fatigue cracks may occur at maximum tensile stress locations in the aircraft structure. The current study about aircraft includes the local and global stress analysis of the fuselage shape near the wing attachment. Here in this paper fatigue and damage tolerance design, analysis, testing and service experience play an important role for the airworthiness of an aircraft during its entire service life. This project also includes fatigue damage calculation for an aircraft load spectrum. In aircraft metallic structure fatigue establishes itself to form a crack, which propagates. Fatigue life calculation also carried out using constant amplitude S- N data is used, if the crack in a critical location leads to a catastrophic failure of aircraft.

Keywords: *Aircraft fuselage, catastrophic failure, fatigue crack, FEA, stress.*

I. INTRODUCTION

The design of an Aircraft is finding the best possible proportion of the mass of the vehicles and pay load. It needs to be strong and stiffing enough to restrict the exceptional circumstances in which Aircraft has to operate and Durability is an also important factor. Also, if any part fails, it does not essentially result in failure of the whole aircraft. The main components of an aircraft are tail, wing and fuselage, for define its exterior shape. The load- bearing associates in above main sectors are subjected to key forces, are called the airframe. The airframe is what residues if all apparatus and structures are coated absent. The Ancient aero planes had skin complete from saturated linen that could hardly communicate some force. The fuselage is the one of the major part of an aircraft or main structure of aircraft. Fuselage provides space for peopl, load controls and some of the equipment. The fuselage is attached with wings, power plant, empennage and landing gear. The fuselage consists of the cockpit and cabin and which has places for passengers and airplane controls. The fuselage gives space for load and main airplane components by attachment points. The type of truss fuselage is fabricated of aluminum tubes stiffness is done by welding the tubes connected into a triangular shape which are called as trusses.

II. LITERATURE REVIEW

Sriranga B.K, Kumar: This paper "Stress analysis and fatigue life prediction of fuselage wing lug attachment bracket of a transport" deals with the bending load transmission combined will be considering for the analysis. Primary one wants to ensure the stationary load resonant ability of the fuselage- attachment with wing. The stress analysis is to be conducted for known geometry of the fuselage-wing attachment bracket. FEM is used for the analysis of stress. In the current project, an attempt will be made to predict the fatigue life of fuselage-wing attachment bracket in a transportation aircraft. In a metallic manufacture fatigue manifests itself in the form of a crack propagates. Fatigue cracks will appear at the location of high tensile stress locations. These locations are invariably of high stress concentration. Fatigue life calculation will be carried out for typical service loading condition using constant amplitude S-N data for various stress ratios and local stress history at stress concentration. In this paper for modeling CATIA V5 software is used and for analysis tool MSC/ PATRAN and MSC/ NASTRAN 2010. **[1]**

Venkatesha B K, Deepak Kumar T & Prashanth K P: This paper "Investigation of Fatigue Crack Growth Rate in Fuselage of Large Transport Aircraft using FEA Approach" focuses its consideration on damage tolerance design of a fuselage assembly of transportation aircraft. The objective of this current paper is to investigate crack initiation and crack growth rate in the flat stiffened panel of fuselage structure. The longitudinal crack is originated from the location of rivet hole and MVCC method is used to calculate SIF through each period of crack**.[2]**

Abbani Rinku, Prashanth R, Naveen Kumar R O: This paper "Structural optimization of typical light transport aircraft components" deals with topology and size optimization of three structural components. Stub-wing spar, Nose landing gear bracket and wing attachment bulkhead of a typical light transport aircraft. By using these techniques weight reduction of 15% to 30% has been achieved. Optimization is registered in 1940. George Dantzig used mathematical methods for making programs for military application. This current paper shows how topology and size optimization tools can be design of aircraft component and this technology can be used in the manufacturing environment with small industrial period scales and which is able to deliver effective strength and stability module design**.[3]**

III. METHODOLOGY

Here the fuselage segment comes up in many design but in this project we particularly considering the attachment brackets keeping in mind of design and weight of the structure. The fuselage segment structure has been modeled in cad (CATIA-V5), Fuselage segment having different parts such as Bulkhead, Longerons, Lugs and Skin. For the same FE model were generated using PATRAN. The steps involved in Finite element models as shown in the Figure 1.

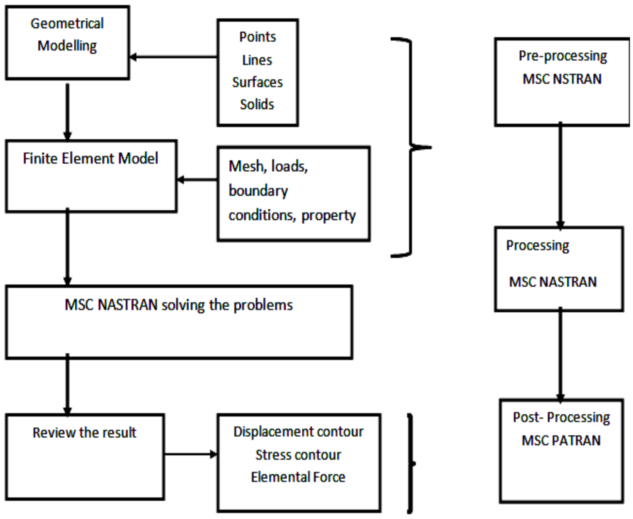

Fig 1: Steps involved in Finite Element Analysis

CAD model was imported in PATRAN and0necessary geometry was done using quick edit panel in PATRAN, The FE fuselage segment depends upon the configuration of geometry elements types are classified as 1D, 2D, 3D. As per our geometry configuration we choose 2D elements (Shell Meshing) for FE modeling. Shell Meshing was carried out using manual Meshing as well as auto Meshing commend. And as per sectional properties in the geometry we assigned sectional properties using card image (sectional shell) and element formed and used to assigned thickness.

Material properties was assigned to the FE Model using Aluminium material in this project and material properties such as Pressure and Stress data, Young's Modulus, Poisson's ratio and Yield Strength is defined and boundary condition was applied as per paper description to the Fuselage segment was fixed in all degree of freedom

These above shown data's are very useful to plot post processing. And NASTRAN solver was exported and fired a run and what the results are getting post processed using Nastran software. Due to increasing cost on conducting real-time aircraft worthiness simulations, here CAE tools are used in automotive and automobile industry. As per needed result, product development cost has less compared to existing ones. The capability of CAE tools in these days has progressed to the point of the design very much and finally validation is done using computer simulations rather than physical testing.

A. Geometrical Configuration

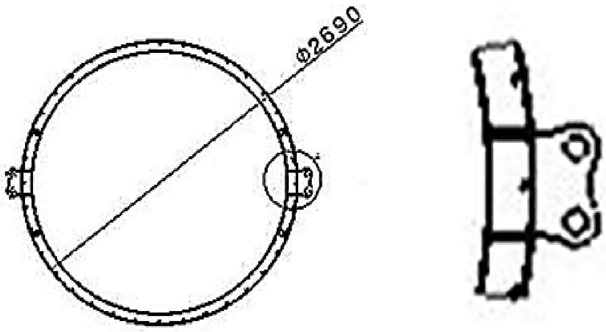

Fig 2: Front view of fuselage lug location

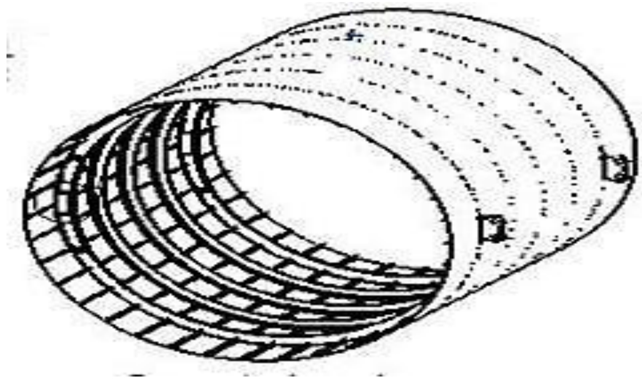

Fig 3: Isometric view of fuselage segment

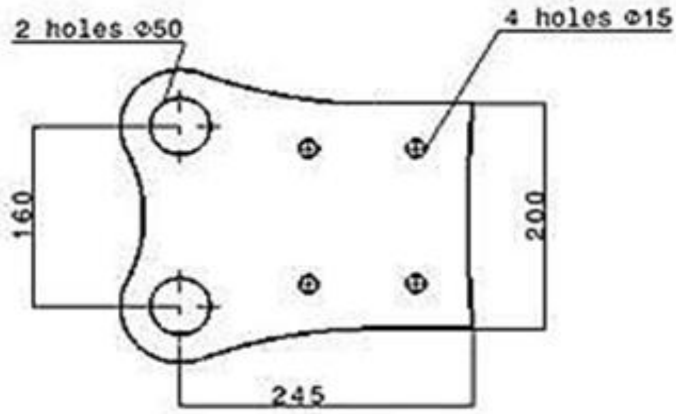

Fig 4: Geometrical details of lug: Front view

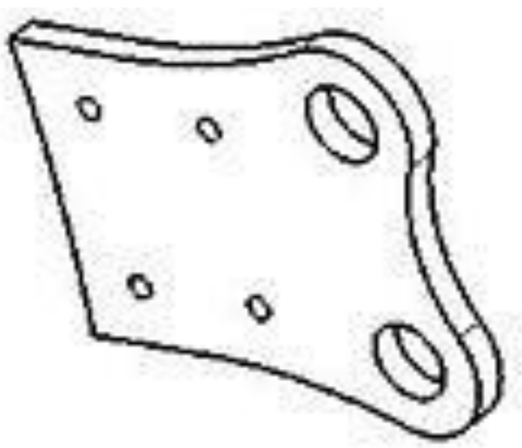

Fig 5: Geometrical details of lug: Isometric view

Table 1: Material property used for analysis

Properties	Aluminium 2014-T6
Density	2.8 g/cm^3
Ultimate tensile strength	483Mpa
Poisson's ratio	0.33
Young's modules	72.4 GPa
Shear strength	290 GPa
Fracture strength	124Mpa
Shear modules	28GPa
Bearing yield strength	662Mpa
Ultimate bearing strength	889Mpa
Elongation at break	13%

IV. STRESS ANALYSIS OF FUSELAGE SEGMENT WITH WING ATTACHMENT BRACKET

A. Bracket

It includes the processing, pre-processing and post-processing phase. Pre-processing phase includes in details of meshing, loads and boundary conditions. These processing carried out by MSC PATRAN. The FE model is development of the structural problem having an element, node position, physical parameters, loads and boundary conditions. The determination of FE model is to create that makes exactly as existing model and makes suitable input files for the variable FE solver. For the formulation of geometrical model the available surface area can be chosen because change of movement and stiffness. Surface mesh is generated by using 2D elements such as quads and trias elements and fine meshing is carried out near the mouse hole cut out to capturing the gradient stress distribution. Quality checks such as aspect ratio, skew, warp, taper ratio are verified and the connectivity elements in the finite element model are checked.

Surface mesh is generated by using 2D elements such as quads and trias elements and fine meshing is carried out near the mouse hole cut out to capturing the gradient stress distribution. Quality checks such as aspect ratio, skew, warp, taper ratio are verified and the connectivity elements in the finite element model are checked. Surface mesh is generated by using 1D element and representation of longeron is done sectional properties and bar orientations are assigned to this 1D element. Surface mesh is created by using 2D elements such as quads and trias elements and fine meshing is carried out near the mouse hole cut out to capturing the gradient stress distribution. Quality checks such as aspect ratio, skew, warp, taper ratio are verified and the connectivity elements in the finite element model are checked.

Skin meshing will be done by surface meshing, it is generated by using 2D elements such as quads and trias elements and fine meshing is carried out near the mouse hole cut out to capturing the gradient stress distribution. Quality checks such as aspect ratio, skew, warp, taper ratio are verified and the connectivity elements in the finite element model are checked. The fuselage segment meshing will be done by surface meshing and it is generated by using 2D elements such as quads, trias and finite mesh is carried out near the mouse hole cut out to capturing the gradient stress distribution. Quality checks such as aspect ratio, skew, warp, taper ratio are verified and the connectivity elements in the FE model are checked. The meshing of fuselage, lug and skin are meshed in 2D. The meshed models are shown in above figure. The rivets are represented by 1D element which connects bulkhead, skin and bolts are represented using 1D element which connects bulkhead and lug.

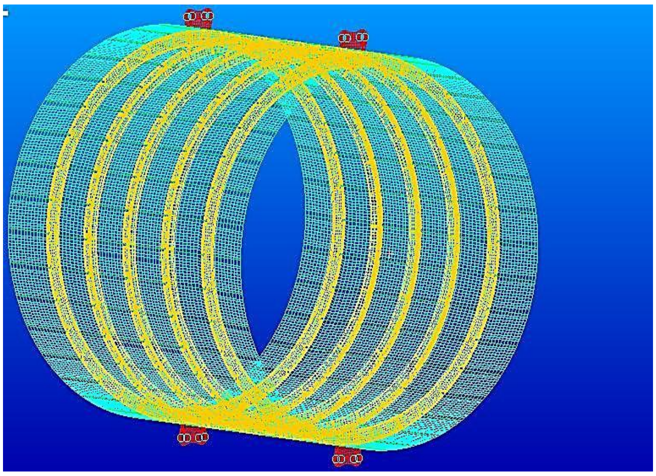

Fig 6: Finite element model of Fuselage segment

V. DATA REDUCTION

Finite Element model considering the boundary and load conditions. A load is applied to the one end of finite element model. This applied load is mainly generated the essential bending moment at the root.

A. Boundary Condition

Weight of the aircraft considered = 2722 Kg
Design load factor = 3G
Total design load on the aircraft = 3×2722 = 8166 Kg
Weight carried by wings = 80% (8166) = 6532 Kg
Weight carried by one wing = 6532/2 = 3266 Kg
Weight carried by front spar = 55% (3266) = 1796 Kg
Weight carried by rear spar = 45% (3266) = 1470 Kg

B. Loads due to cabin pressurization

Stress analysis of rear pressure bulkhead at different load condition starting from 6 psi to 9 psi was done with NASTRAN software.

$$\sigma_h = \frac{p \times r}{t} \quad (1)$$

Where
σ_h=hoop stress in Kg/mm^2
p = pressure in Kg/mm^2
r = radius of fuselage in mm
t = thickness in mm
p = 6 psi, 1 psi = 0.0007 Kg/mm^2
p = 6×0.0007 = 0.0042 Kg/mm^2
r = 1345 mm
t = 1.5 mm

$$\sigma_h = \frac{0.0042 \times 1345}{1.5} = 3.766 kg / mm^2$$

VI. RESULTS AND DISCUSSIONS

A. Loads due to Cabin Pressurization

The following Figure 7, show that loads and boundary condition of fuselage segment due to cabin pressurization.

The one end of the fuselage segment is fixed and pressure is applied to the entire model. The pressure is circulating inside the fuselage segment through the rivets.

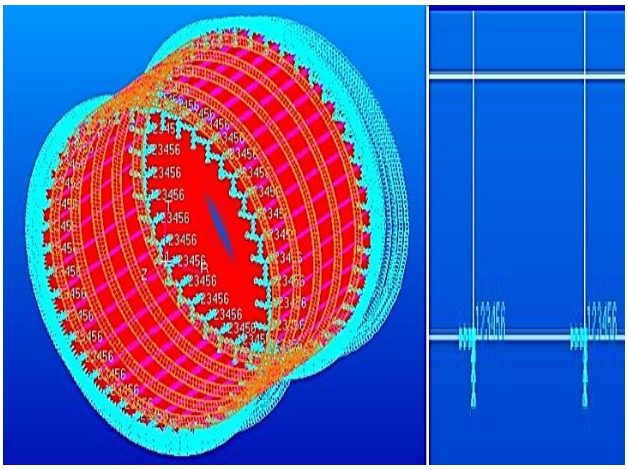

Fig 7: Finite element model of Loads and boundary condition due to cabin pressurization

B. Deformation and Stress

The fallowing Figure 8 and 9 shows stress contour of fuselage segment due to cabin pressurization. In this model we obtain the maximum stress location near the bulkhead hole. The magnitude of maximum stress is found to be 10 Kg/mm^2 near the mouse hole cut-out of the bulkhead.

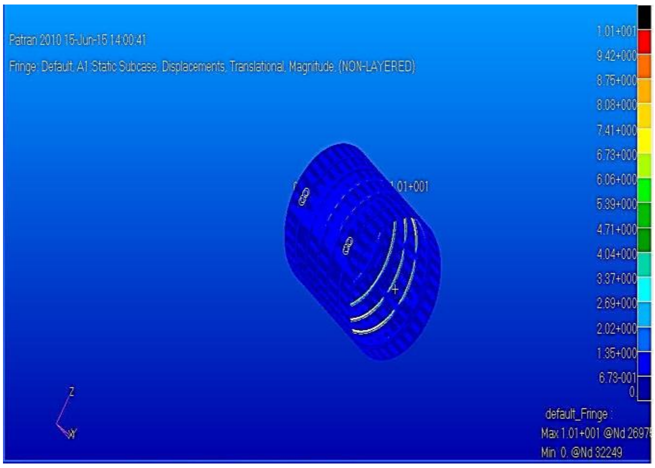

Fig 8: Deformation contour due to cabin pressurization

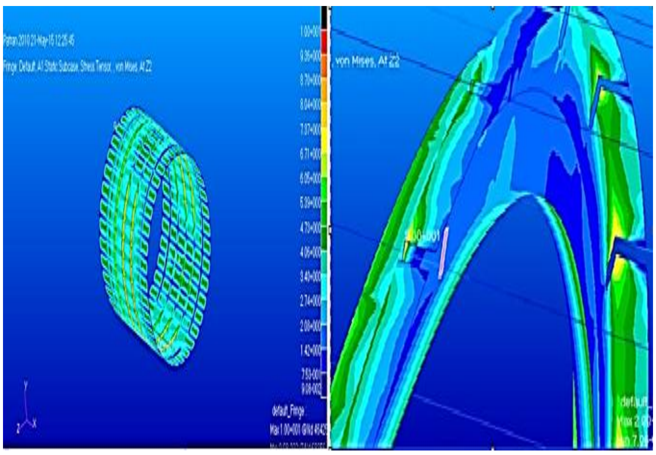

Fig 9: Stress contour of the bulkhead of the fuselage segment due to cabin pressurization

C. Load due to Wing Bending

The following Figure 10 shows loads and boundary conditions of fuselage segment due to wing bending. The one end of the fuselage segment is fixed and load applied on the other end. The load is also applied on the lugs of both front end and rear end.

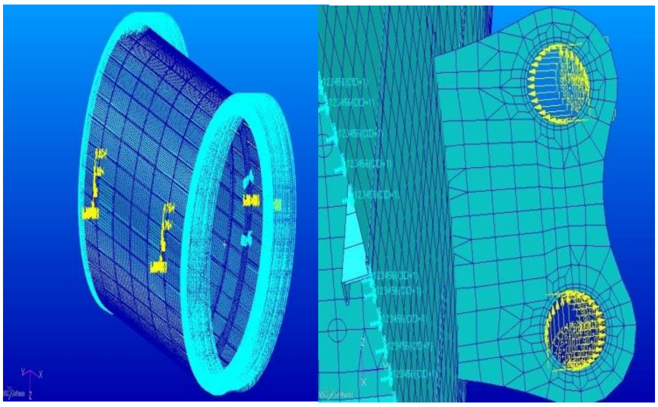

Fig 10: Loads and boundary conditions due to wing bending

D. Deformation and Stress

This following Figure 11 shows the Displacement contour of fuselage segment. The maximum displacement is 10.5 mm and minimum displacement is 0.697 mm

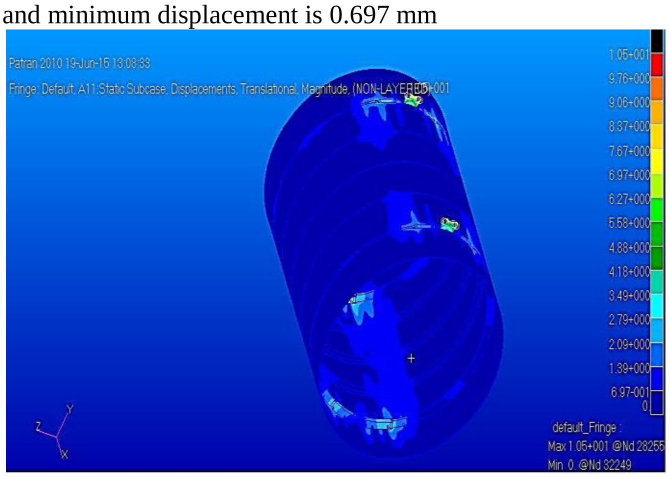

Fig 11: Deformation contour due to wing bending

The below Figure 12 shows stress contour of fuselage segment due to load. In this model we obtain the maximum stress location in the lug portion. The magnitude of maximum stress is found to be 37.5 Kg/mm^2 near the pin hole of the attachment bracket.

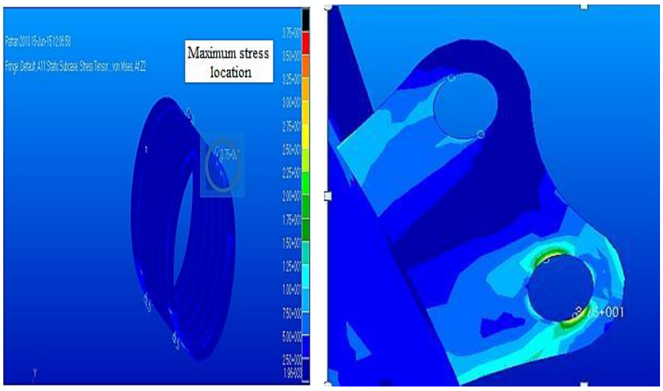

Fig 12: Stress contour of wing fuselage attachment bracket

VII. CONCLUSIONS

1. Stress analysis of fuselage segment with wing attachment bracket under combination of cabin pressurization and lift component of aerodynamic force was done using analysis tools MSC Patran and MSC Nastran.
2. Maximum Tensile stress of magnitude 38.1 kg/mm^2 was observed at the pin hole location of the attachment bracket under combined loading , which was less than yield strength of Al 2014 T6 alloy.
3. Fatigue damage is calculated for the maximum tensile stress location (pin hole location) using Minor's Rule and damage is found to be zero.
4. The structure is safe and there is no crack initiation in the structure throughout its service life.

VIII. Future Scope

1. Failure of fatigue crack growth analysis may be determined on the wing fuselage lug bracket in the bottom skin of the wing.
2. Fatigue damage evaluation for the lug may be determined for given load spectrum.

References

[1] SrirangaB.K1, Kumar.R2 : "Stress Analysis and fatigue life prediction of fuselage wing lug attachment bracket of a transport" IJRET: International Journal of Research in Engineering and Technology ISSN: 2319-1163 / ISSN: 2321-730 [2] Venkatesha B K, Deepak Kumar T & Prashanth K P: "Investigate of Fatigue Crack Growth Rate in Fuselage of Large Transport Aircraft using FEA approach" Global journal of Research in Engineering Mechanical and Mechanical engineering volume 2024, Type: Double Blind peer Reviewed international journal paper.

[3] Abbani Rinku, Prashanth R, Naveen Kumar R O: " Structural Optimization of typical light transport aircraft components" Scientist in C-CAAD, NAL, Bangalore, Scientist in C-CAAD, NAL Bangalore, Design Engineer ADES Pvt. Ltd.

[4] Karthik N1, Dr. C Anil Kumar: " Stress Analysis for a Wing Attachment Bracket of a six seater Transport Aircraft Structure" International Journal of Innovative Research in Science, Engineering and Technology, Vol.2, Issue 7, July 2013.

[5] J.J.M. de Rijck, J. Schijve, S.A. Fawaz, J.J. Homan and R. Benedictus3: "Stress Analysis of Mechanically fastened joints in aircraft fuselage" 24th ICAF Symposium Naples, 16-18 May 2007.

[6] Sreyas Krishnan SI, Anish R, Girish K E, " Stress analysis of the back pressure bulkhead of a fatigue life estimation and fuselage structure" Department of Mechanical Engineering, SAINTGITS College of Engineering, Kottukulam Hills, pathamuttam, Kottayam 686532

[7] M R Urban: "Study of Fatigue Life of a Riveted metal sheet of aircraft connections" structures Research Development, Sikorsky Aircraft Corporation, 6900 Mian Street, P.O. Box 9729, Stratford, CT 06615-9129, USA 2003.

[8] J C. Newman, R.S. Piascik, Jr C.E. Harris and D.S. Dawicke: " Methodology for predictiong the onset of widespread fatigue damage in lap-spice joints" NASA Langley Research Centre Hampton, Virginia USA December 1998.

[9] Shamsuzuha Habeeb, and K.S. raju: " Crack Arrest Capabilities in Adhesively Bonded Skin and Stiffener", Proceedings of the 5th Annual GRASP Symposium, Wichita State University, Volume 16, Issue 6, PP. 620-657, 2009.

[10] Frost NE, and Dugdale DS: " The Propogation of Fatigue Cracks in Test Specimen", J Mech phys Solids Vol. 6, PP. 92-110, 1958.

[11] Newman Jr JC, Brot A, and Matias C: " Crack growth calculations in 7075-T7351 aluminium alloy under various load spectra using an improved crack- closure model", Engineering Fracture Mechanics, pp. 71.2347-63, 2004.

[12] L. Molent, R. Jones, S. Barter, and S. Pitt: " Recent Development in fatigue crack growth assessment", International Journal of fatigue, Vol.28, pp 1759.

[13] Gere, J.M. and Timoshenko, S.P.: " (1991), Mechanics of material, Chapman and Hall, London.

[14] Schijve, J. : (1972) some elementary Calculation on Secondary Bending in Simple Lap Joints, NLR-TR-72036, Amsterdam, National Aerospace Laboratory.

AUTHORS PROFILE

Prof. Praveenkumara B M obtained his bachelor's degree in Mechanical Engineering from PES College of Engineering Mandya, Karnataka, India and Master Degree in Thermal Power Engineering from VTU PG Center Gulbarga. Currently he is working as Assistant professor in the department of Mechanical Engineering Vidyavardhaka college of Engineering Mysuru Karnataka India, Since July 2019. His research interest is on Aerodynamics, Heat exchanger and Refrigeration.

Prof. Bhanuprakash M J obtained his bachelor's degree in Mechanical Engineering from GEC Kushalnagar, Karnataka, India and Master Degree in Machine design from VVCE, Mysuru. Currently he is working as Assistant professor in the department of Mechanical Engineering Vidyavardhaka college of Engineering Mysuru Karnataka India, Since July 2019. His research interest is on CFD, Aerodynamics and Composite Materials

Prof. Rajesh Kumbara S K obtained his bachelor's degree in Mechanical Engineering from JNN College of Engineering Shivamogga, Karnataka, India and Master Degree in Computer Integrated Manufacturing from Sir. MVIT Bengaluru. Currently he is working as Assistant professor in the department of Mechanical Engineering Vidyavardhaka college of Engineering Mysuru Karnataka India, Since July 2019. His research interest is on Aerodynamics, Industrial Automation and Robotics.

Dr. B Sadashivegowda obtained his bachelor's degree in Mechanical Engineering from AIT, Chickmagalur, Karnataka, India and Master Degree in Refrigeration and Air Conditioning from CIT Coimbatore Tamilnadu India. Later he obtained his doctral degree from prestigeous Indian Institue of Science Bangalore India, in the field of refrigeration. He is serving as Principal and Professor in the department of Mechanical Engineering Vidyavardhaka college of Engineering Mysuru Karnataka India, Since June 2007. Before joining Vidyavardhaka college of Engineering he served as Head in Department of Engineering and natural sciences, American Degree Program Taylor's College, Subang Jaya Selangor Darul Ehsan, Malaysia from 1999 to 2007. He has about 32 years of teaching experience. His research interest is Refrigeration and Air conditioning, Alternate fuels and IC Engines.