ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Fatigue Life Estimation of Butt Joints on a Aircraft Wing

Sheldon D'souza
M.Tech (Machine Design)
Department of Mechanical Engineering
Malla Reddy College of Engineering and Technology.

Mr.K.Bitcha, M Tech
Assistant Professor
Department of Mechanical Engineering
Malla Reddy College of Engineering and Technology.

ABSTRACT

Commercial aircrafts or corporate aircrafts are a need of an hour today for mankind in order to do their things faster and reach different destinations sooner than ever. The aircraft companies manufacturing these aircrafts have to fulfill the demands of the airliners and fulfill their requirements and see that they also make a profit. The aircrafts made by them have to be highly reliable, durable, light weight and be airworthy as lot of lives are at stake once people are on it. The manufacturers are on a very tight schedule as they have to break even and also meet the demand.

During the service life a regular aircraft is subjected to different kinds of loads, such as thrust loads, inertia loads, temperature, gusts, hail storms and much more. The combination of these loads has to be taken into account to structurally analyze the different components of the aircraft. The complex geometry of the aircraft makes the job much more difficult and tools like FEM backed up by extensive programs have to be used in order to perform such simulations.

The focus of this project is to estimate the design limits and perform damage tolerance on the upper surface of the wing. The stresses over the model are computed and made use of to perform fatigue analysis.SN curves and miners rule is made use of to get the fatigue life of the component and later part is completed by using fracture mechanics approach. Due to the presence of High Tensile stresses a crack can be predicted at one of the bolt joints of higher diameter which would lead to catastrophic failure. So a finite element tool is used to simulate this process such that the crack phenomena can be pre-determined and to estimate the stress intensity factor and stresses.

The driving force is compared with the resistance curve to estimate the crack growth such as to assess the fatigue strength of the component and stay on course with the life of the aircraft

INTRODUCTION

In aircraft structures, lap joints and butt joints are widely used for aircraft construction. The major function of the joints carrying skin is to carry the loads and distribute it along the ribs and other secondary structures of the aircraft. Aircraft body is subjected to various temperature gradients and is also subjected to high gust loads in its life cycle. The service life of critical aerospace structures is governed by failure modes such as: fatigue, fracture, yielding, creep, corrosion, erosion, wear, etc. The airplane lap and butt joints are one of the critical structure components which must endure substantial mechanical loading. High cycle Fatigue (HCF) life estimation of the joints is therefore very important in order to declare a safe operating duration before which the joints are retired from service. Stress analysis of the joint section of a developmental aircraft is carried out in this work. Stress analysis results shall be used for the development of a generic HCF life model for the butt joint.

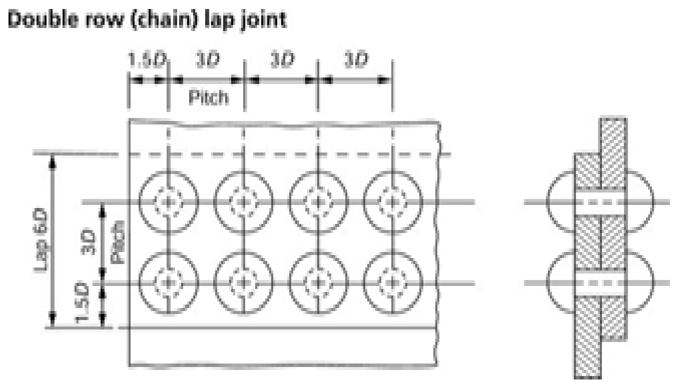

Figure 0-1 : Lap joint top view and cross-section

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Wings

A complete airplane structure is manufactured from many parts. These parts are made from sheets, extruded sections, forgings, castings, lubes, or machined shapes, which mustbe joined together to form subassemblies. The subassemblies must then be joined together to form larger assemblies and then finally assembled into a completed airplane. Many parts of the completed airplane must be arranged so that they can be disassembled for shipping, inspection, repair, or replacement, and are usually joined by bolts or rivets In order to facilitate the assembly and disassembly of the airplane. The basic functions of an aircraft's structure are to transmit and resist the applied loads; to provide an aerodynamic shape and to protect passengers, payload systems, etc., from the environmental conditions encountered in flight. These requirements, in most aircraft, result in thin shell structures where the outer surface or skin of the shell is usually supported by longitudinal stiffening members and transverse frames to enable it to resist bending, compressive and torsional loads without buckling. Such structures are known as semi-monocoque, while thin shells which rely entirely on their skins for their capacity to resist loads are referred to as monocoque

Objective of the work

To carry out stress analysis of a Wing box in order to develop a generic life model for Fatigue life estimation of the butt joint under any operating condition (combination of loads and temperature). During operation, the wing is subjected to a combination of loads, viz. lift loads, forces due to inertia and thermal stresses due to the temperature differences in the nature. Detailed stress analysis is carried with the above loads for the wing box to arrive at the stress state in the skin and to identify the critical locations in the wing box. From the stress analysis results, S n curve based approach shall be used to establish the Fatigue life of the disc. The procedure of stress analysis followed by Fatigue life estimation shall be repeated for a series of load cases to arrive at the generic Fatigue life model and there after proceed with damage tolerance.

LITERATURE REVIEW

This part provides a review of the previous work done in the broad area on the finite element modeling, stress analysis and life estimation for aero engine discs.

Brief Review of Literature Survey

This part of the chapter provides the reviews of the previous works done on the Fracture and Fatigue analysis. The literature review is set in such a format that the basic need for the present work and developments obtained in the field of research to accomplish the efficient method of elastic analysis.

S.A. Zamani[1]

In this work, analytical and FE analysis is performed of an annular disc that results show how the centrifugal force affect the rotating annular disc performance and effect of Poisson's ratios on Von-Mises stress, Radial and tangential strains, radial displacement. In the axis-symmetric case the radial stress distribution around axial only and the radial displacement is vary along the radial distance where the circumferential (hoop) displacement taken zero. Investigation found in a similar radial distance, the radial displacement of a rotating annular disc increases with decreases Poisson's ratios. The most important investigation of this study is the assumption of small deformation at high rotating speed not applicable. The finite element software gives good approximate result for large deformation.

The assumption of large deformation for high rotating speed is acceptable. As the Poisson's ratio decreases the radial displacement, Von-Mises stress and radial strain increases. The finite element analysis gives acceptable result of a rotating annular disc at high rotating speed.

MODELING

Chemical composition of Al alloy

COMPONENT	Wt%
Al 90.7-94.7	90.7-94.7
Cr Max 0.1	0.1
Cu 3.8-4.9	3.8-4.9
Fe Max 0.5	0.5
Mg 1.2-1.8	1.2-1.8
Mn 0.3-0.9	0.3-0.9
Other, each Max 0.05	0.05
Other, total Max 0.15	0.15
Si Max 0.5	0.5
Ti Max 0.15	0.15
Al 90.7-94.7	90.7-94.7

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Table 0-1Catia Dimensioning

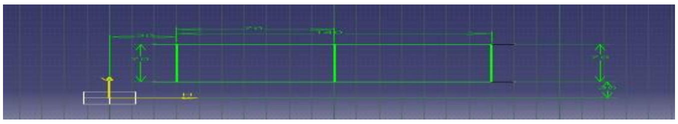

Figure 0-1Dimensions of the model used

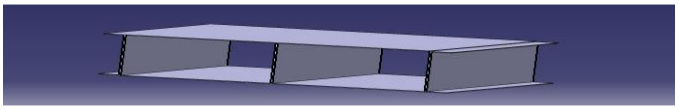

Figure 0-23D view of the wing box

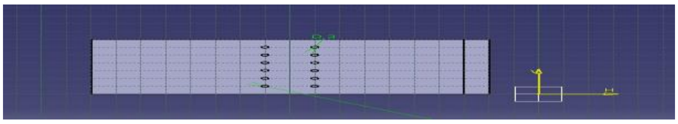

Figure 0-3Top view of the wing box

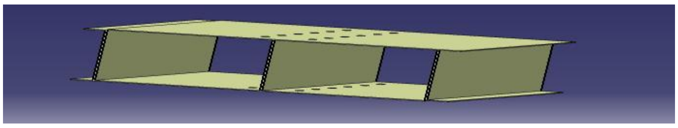

Figure 0-43D model showing riveted holes on the aircrafts

Finite Element Model

The components of wing box such as spar beams, ribs, stiffeners and skin are meshed separately.

Quadratic and triangular elements have been used in this model. The beam elements were used for rivet connections. Near the splice region the fine mesh has been done and the coarse mesh has been done for the rest of the portions of wing box.

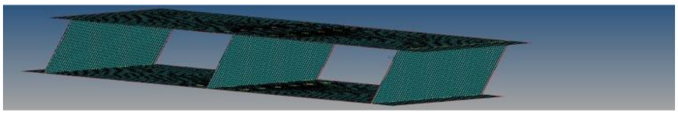

Figure 0-1Meshed model of the inside structure

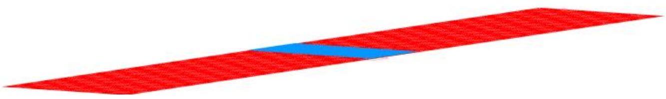

Figure 0-2Hole model for local analysis

STRUCTURAL ANALYSIS

ANSYS

ANSYS is a general purpose finite element modeling and analysis package for solving a wide variety of mechanical problems. These problems have included linear and nonlinear structural analysis, fatigue analysis, buckling, modeling analysis, heat transfer analysis. [13]

The analysis procedure in ANSYS is divided into three major parts

- Pre-processing
- Processing
- Post-processing

Pre-Processing

Here the entire model of the structure is created and the material properties, loads and the constraints are specified. Input to Finite Element Model is provided here.

- **Geometry**:the geometric model is a description of the form of the object to be modeled and some properties that are directly related to the object.
- **Meshing:** It is one of the basic activities they help in converting the constructed geometry into nodes and elements. It involves discretization of the geometric domain into valid zones for analysis. Thus meshing produces a discretized domain that becomes the starting point for the assignment and computations of different analysis quantities. Mashing essentially converts the geometric definition in the problem space into a representation on for a numerical solution in the solution space.
- **Adding Boundary conditions:** Assigning boundary conditions in case of static analysis involves constraining linear and rotational degrees of freedom of relevant nodes and the application of forces, moments, pressures etc., at the relevant nodes.

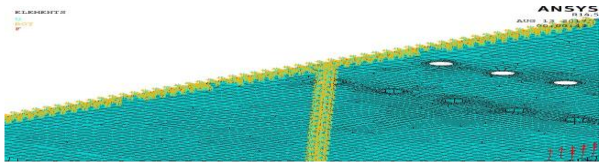

Figure 0-1Boundary conditions applied to the model

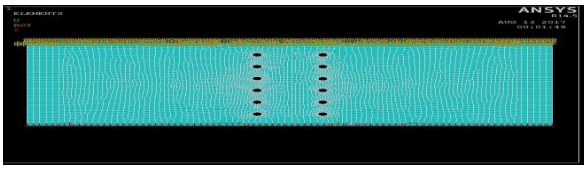

Figure 0-2Top view of the fixed constraints applied on fixed end

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

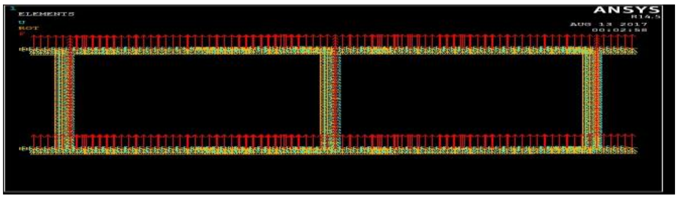

Figure 0-3 Global analysis

The loads and boundary conditions applied to the wing box (global analysis) are shown above

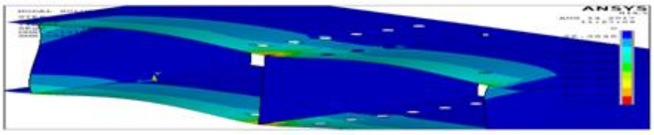

Figure 0-4Stress plot showing the stresses at the joints

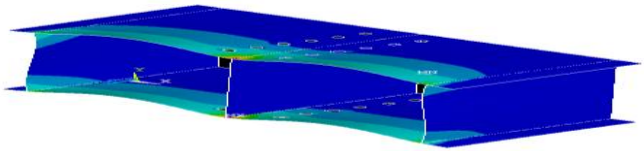

Figure 0-53d view of the entire von mises stress plot

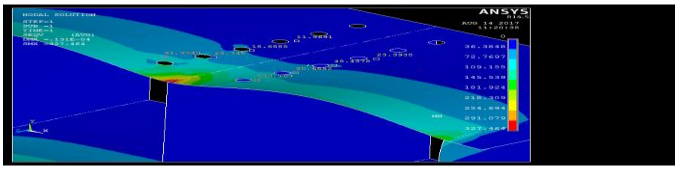

Figure 0-6Stress numbering shown in the above plot

Average Stress value at a distance 187.5mm 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 2.1255

Hence total load = Stress * area = 743.944 kg (1g condition)

For 3g condition, load = 2231.83 kg

Distributed load on cross section for local analysis = 12.75 kg/mm

Averaging all the stress values a stress value of 35Mpa is used for local analysis

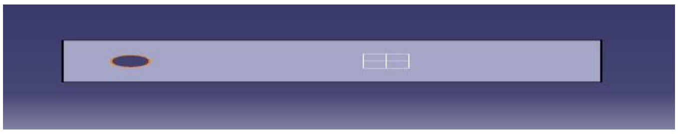

Figure 0-7Catia Model of Local Analysis

Average Stress value at a distance 527.5mm from the spar tip is taken where maximum stress is identified in local analysis for calculation of load in local analysis and which is found to be 3.062

Hence total load = Stress * area = 153.1 kg

Distributed load on cross section for local analysis = 6.124 kg/mm

The loads and boundary conditions applied to the above model are shown below

Stress Contour of the Loaded Hole Local Analysis

Figure 0-8The loads and boundary conditions applied to the above model are shown below

CONCLUSION

- Design and analysis of panel with circular holes was carried out by using FEA software Ansys. The project investigated the stress analysis and fatigue crack propagation and fatigue life estimation of panel with circular holes. The stress analysis gives the visualization of stress concentration near the edge of holes where the crack propagation initiated.
- The crack length at the initial is very small about 1-5 micro mm. The CAE software does not support the very small crack length. Hence, the convectional fatigue life approach (Stress Life approach or high cycle fatigue) method is used to calculate initial fatigue life.
- The Ansys software is used to find stress intensity factor at different crack length (0.39 to 18mm) and crack simulation. The centre crack plate is used to form initial crack in the edge of circular holes of skin panel which gives more accurate result than a rectangular or square crack plate. Number of fatigue analysis is performed to get number of cycles at different crack length

and at different loading conditions.The results got in FE fatigue analysis is approximate which had validated the experiment result obtained in Rig tests.

- ANSYS-14.5 was used to perform fatigue crack growth and estimate number of cycle to fail the holes in joint panel. The finite element model was modeled in CATIA V5 R21 and meshed in HYPERMESH-10. The element type is plane 42 and the total number of nodes 60123 and elements 50112.
- Hence it can be concluded that the obtained results from Ansys 14.5 are well within the acceptable limit to that of the results obtained from Rig tests.

Scope for Future Work

This project presents the design and finite elemental analysis of a skin joint panel with circular holes at different loads. Based on this context various other explorations can be suggested.

- Stress and fatigue analysis of a panel for different materials and different loading condition can be determined.
- An analysis of the literature shown that the most of investigations of small crack behavior in aircraft skin panel with holes alloys were performed at roomtemperature. Future fatigue crack growth studies should be performed at relevant temperatures. Finite elemental analysis can be carried out for the panel to determine the Fatigue behavior based on temperature.
- Future fatigue crack growth studies should be performed at panel relevant plasticity consideration that is EPFM.

References

1. Deb K, Multi-objective optimization using evolutionary algorithms, 1st ed. Wiley, New York, 2001.

For Journal / Proceedings
Examples:

2. J. Clerk Maxwell, A Treatise on Electricity and Magnetism, 3rd ed., vol. 2. Oxford: Clarendon, 1892, pp.68-73.
3. I.S. Jacobs and C.P. Bean, “Fine particles, thin films and exchange anisotropy,” in Magnetism, vol. III, G.T. Rado and H. Suhl, Eds. New York: Academic, 1963, pp. 271-350.
4. K. Elissa, “Title of paper if known,” unpublished.
5. Robert D Cook “Finite element modeling for stress analysis” John Wiley & Sons, 1st edition
6. S S Rao“Finite element method in engineering” Elseveir Publications, 4th edition
7. T H G Megson “Aircraft structures for engineering students”Elsevier Publications, 4th edition
8. Analysis of the wing box with spliced skin and estimation of fatigue life for the wing box’ by S sarath