Stress Analysis of the Wing Box with Spliced Skin

Vijay Praveen P M[1] Vijaya Kumar S[2], Hareesh N S[3]
Mechanical Department,R L Jalappa Institute of Technology,Doddaballapur, india
Mechanical Department, REVA Institute of Technology & Management,,Bangalore, india
Mechanical Department, REVA University,,Bangalore, india

***Abstract*— Aircraft is symbol of a high performance mechanical structure with a very high structural safety record. Safety and the structural weight are important parameters to be considered in the design phase. Rarely an aircraft will fail due to a static overload during its service life. For the continued airworthiness of an aircraft during its entire economic service life, fatigue and damage tolerance design, analysis, testing and service experience correlation play a pivotal role. 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 [10]. 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.**

Keywords: Aircraft, Wing box, Spliced skin, Stress analysis, FEM

I.INTRODUCTION

The wing box 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 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 wing box 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 impact sustained during like turbulence to keep the plane in the air. In a wing box, 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 flexibility 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.

II. 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 wing box to compute the stresses at rivet holes due to tension with the help of MSC PATRAN and MSC NASTRAN.

Al 2024-T351 is used in current wing box due to high strength and fatigue resistance properties. The ultimate tensile strength of this material is 485 MPa and yield strength is 280 MPa)and it has an elongation of 19% (Michael, 1993).

III.METHODOLGY

The following detailed methodology is adopted to meet the desired objective as shown in fig 1

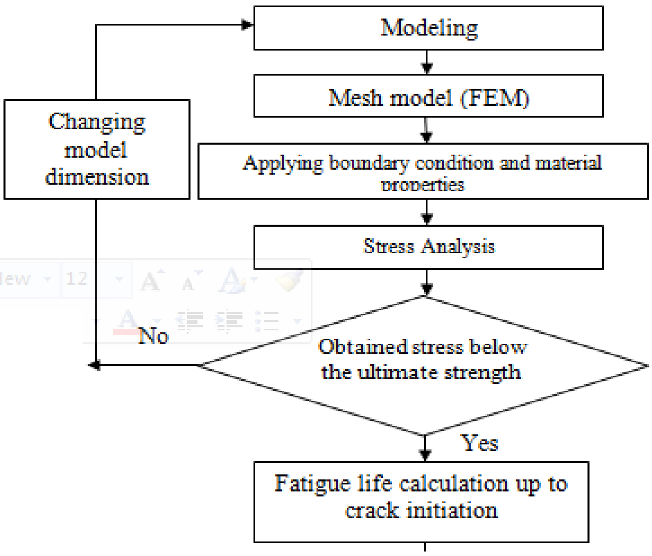

Fig.1. Methodology flow chart

IV.GEOMETRICAL CONFIGURATION

Wing box modeled in CATIA was been shown in fig 2. It consists of different structures. Wing box used here consists of five ribs including a middle rib, stiffeners, bottom and top skins, spars. Each part is modeled in CATIA software and assembled to form wing box.

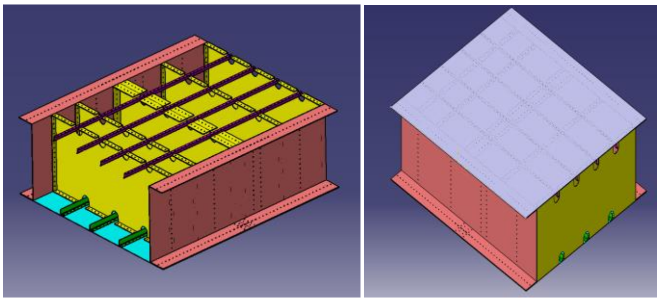

Fig.2. CAD model of the wing box

V.FINITE ELEMENT MODEL

FE model of the wing box is done by using CQUAD and CTRIA shell elements as shown in fig 3 using PATRAN FEA package. To provide connectivity CTRIA shell elements are used. CTRIA elements are also used for decreases the element at flat surfaces and also the CTRIA element are used at irregular shapes

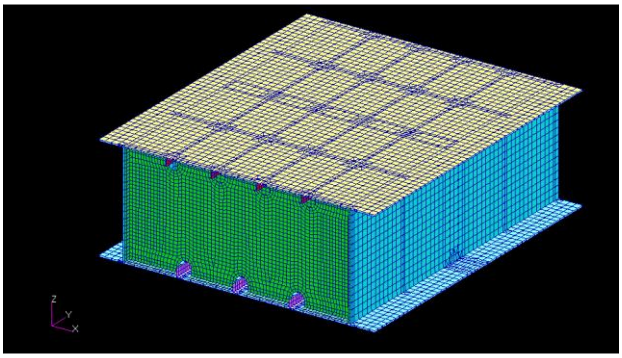

Fig3. Finite Element Model of the Wing Box

VI. LOAD & BOUNDARY CONDITION

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 maximum load is acted nearer to the wing roots and minimum load is acted at the tip of a wing box.

Weight of the aircraft: 44145 N
Design load factor: 3"g”
Factor of safety: 1.5
Therefore, Total design load on the aircraft will be: 19865 N
As 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 = 158922 N
Therefore total load acting on the each wing = 7946 N

But the resultant load is acting at the distance 9000 mm from the wing root as shown in fig 4.
Bending moment at the root of the wing can be calculated as $71.514*10^6$ Nmm
load required at section C-C to simulate the ate the bending moment is P =58141N.
Load distributed on the cross section =19.328 N/mm.
The wing subjected to UDL therefore this load acting at one end of the wing box simulate like cantilever beam shown in fig 4

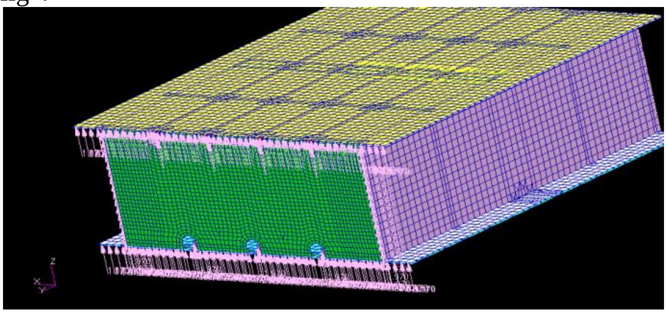

Fig.4. UDL 19.328 N/mm applied at one end of the wing box

All degree of freedom is constrained at end of the wing box connected to fuselage (T_X=Ty=Tz=0, Rx=Ry=Rz=0) as shown in fig 5

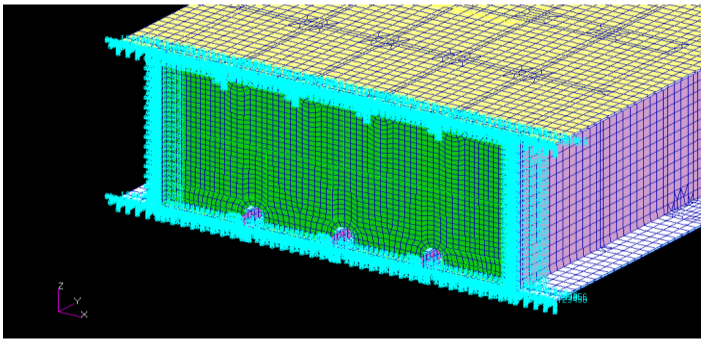

Fig.5. All degree of freedom is constrained at one end of wingbox

In real scenario rivets head resist rotation in X direction due to bending therefore rotation about X direction has to be constrained i.e. (R_x= 0) as shown in fig6.

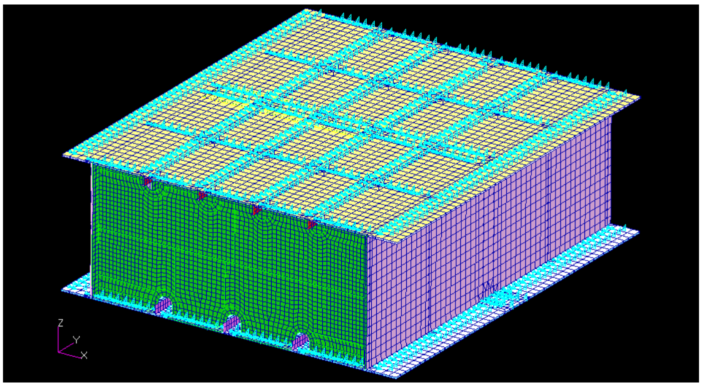

Fig.6. Rx=0 is constrained for rivets

VII. RESULTS

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 fig 7. The magnitude of maximum principal stress developed here is 131.45 N/mm^2. By 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 121.12 N/mm^2.

Since the maximum stress occurred at same rivet location therefore for the same location local analysis on the carried out. The uniform stress regions around the maximum stress concentrated locations of the splice joint are identified from

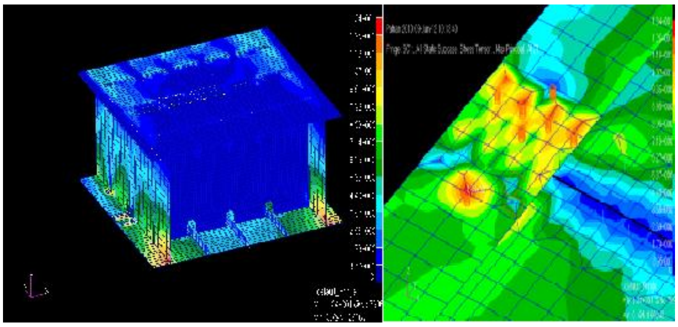

Fig.7. Max Principle stress at bottom rivit location.

global analysis of wing box. It's then meshed separately forming different groups shown in fig 8.

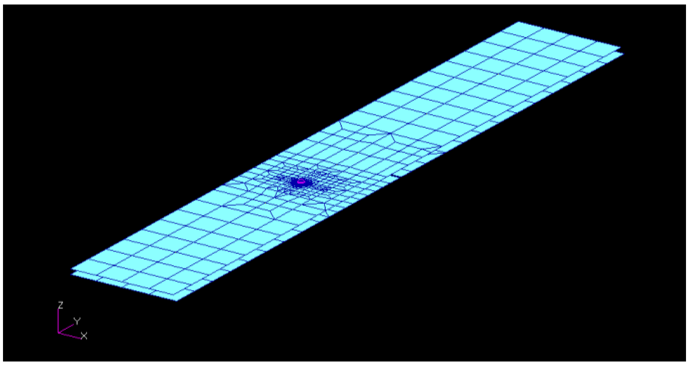

Fig.8 FEM Geometrical Configuration for the Local Model

In local analysis the rivet hole can be simulated at max stress rivet location. Using MPC RBE2 at rivet hole for ensure the connectivity & load transfer. Convergence study is carried out for different meshing density(0.97,0.49,0.32 & 0.24mm).For element size 0.24mm the result is converges i.e 251.13 N/mm^2 exact value of max Principle is obtained as shown in fig 9

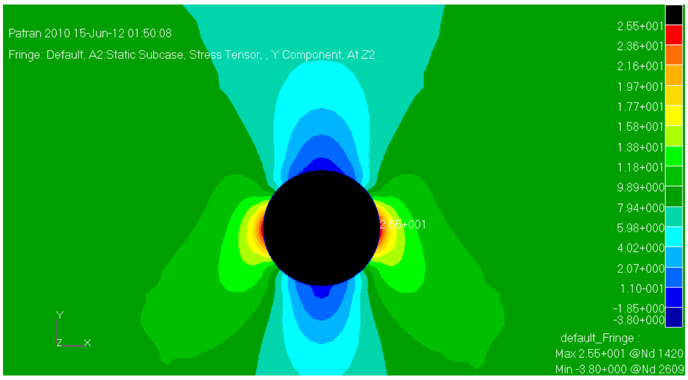

Fig.9. Max stress 251.13N/mm^2 is developed near rivet location for element length 0.24mm

VIII. CONCLUSION

Stress analysis of the wing box 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 251.13 N/mm^2 is observed in the wing box.

REFERENCES

[1] Composite Aircraft Wing box Structure", journal of engg,vol 6,Dec 17 2011.

[2] RamzyzanRamly et al,"Design and Analysis for Development of a Wing Box Static Test Rig", International conference on science and social research, 2010,pp. 113-117.

[3] Stevan Maksimovic, "Fatigue Life Analysis of Aircraft Structural Components",Scientific-Technical Review, Vol.4, No.1, 2005.

[4] N.W.M. Bishop et al, "Finite element based fatigue calculations", Published by International association for the Engineering analysis community, 2000, pages 23-44.

[5] Cui Jianguo et al , 'The Fatigue Life Prediction Model Research for Aero plane Structures', Aeronautic Science Foundation of China,2000, pp. 686-690.

[6] M.D. Halliday, C. Cooper, P. Poole and P. Bowen, "On predicting small fatigue crack growth and fatigue life from long crack data in 2024 aluminium alloy", International Journal of Fatigue, Vol.25, 2003, P pages 709–718.

[7] A.K. Vasudevan, K. Sadananda and G. Glinka, "Critical parameters for fatigue damage", International Journal of Fatigue, Vol.23, 2001, PP. 539–553.

[8] Polagangu James, Gaddikeri Kotresh & D. Murali Krishna, "Design of spar splice joints in composite wing structures", International Conference on Aerospace Science and Technology2, Bangalore, 26 june2008, pages 1-4.

[9] Al.Th. Kermanidis, P.V. Petroyiannis,& Sp.G. Pantelakis, "Fatigue and damage tolerance behaviour of corroded 2024 T351 aircraft aluminium alloy",intenational journal published in 28 jan 2005. pages 1-5.

[10] Jaap Schijve, "Fatigue damage in aircraft structures, not wanted, but tolerated?" International journal of fatigue, Volume 31, Issue 6, June 2009, Pages 998-1011.

[11] Youhong Zhang et al (2009) "Fatigue life prediction of served aircraft aluminum alloy structure", International Conference on Education Technology, pages 313-315

[12] G. S. Campbell and R. Lahey "A survey of serious aircraft accidents involving fatigue fracture", International Journal of Fatigue Vol 6 No 1 January 1984, ISBN 0142-1123/84/010025-06