Stress analysis of Al2014-T6 riveted splice joint fuselage structure using finite element analysis

D Bino Prince Raja1,*, Srinivas Arvind2, Nikita Prashant Vinchure2, G Ramanan3, G Diju Samuel4

1 Aeronautical Engineering, SJC Institute of Technology, Bangalore, Karnataka, 562101, India
2 Aeronautical Engineering, Sha Shib College of Engineering, Bangalore, 562101, India
3 Aerospace Engineering, ACS College of Engineering, Bangalore, Karnataka, 570054, India
4 Aeronautical Engineering, MLR Institute of Technology, Dundigal, Hyderabad, 500043, India

*E-mail: binoaero87@gmail.com

Abstract. This work describes about the stress of a splice joint in a medium transport aircraft. Typical splice joint panel forming part of a fuselage, consisting of skin plates are riveted is considered for the study. The calculations have been performed by considering the cabin pressurization load on the panel. Aluminium alloy 2014 - T6 material considered for structural elements of the panel. Two dimensional finite element analyses have been carried out on the splice joint panel. The distribution of fasteners loads and local stress field at rivet locations read using element analysis. The global finite element analysis linear static analysis first segment of typical fuselage will be carried out. The Global finite element analysis results will be benchmark for comparing the result from the splice joint panel analysis. Repeated finite element analysis will be carried out to get the response of the parent structure fuselage find location the maximum stress location will be identified from the analysis.

Keywords — Stress analysis, Fuselage structure, Al alloy, Finite element analysis

1. Introduction

Aircraft is a complex structure, but a very efficient man made flying machine. The weight of the structure is one of the important factors in the design and development of the aircraft. The orthogonally stiffened skin covers the internal structural elements over the entire aircraft [1]. The many different structural elements of metallic airframe are mechanically fastened to get an integral structural form. In general joints can be designed to be stronger than the members being joined under static loading conditions [2]. On the other hand joints are the most critical fatigue cracking locations in a structure under fatigue loading. This is due to stress concentration, load eccentricity, fretting and other factors present in a joint [3]. The most critical fatigue loading for the fuselage of a transport aircraft is the cabin pressurization cycle which is one cycle per flight. The combat aircraft accident and many other aircraft accidents were due to fuselage fatigue cracking [4]. The problem can be studied through analysis and experimental means by considering a strip of the fuselage skin encompassing the single strap joint. This strip is a free-body containing the riveted joint. Calculations will be performed by considering the cabin pressurization load on the panel [5]. However, in another test program
Jackson review involving threaded bolts; it was found that bolt fit had very little effect on fatigue life [6]. Even increasing the bolt torque in these joints, which is increased the clamp-up and friction forces, did not affect the fatigue lives. Epstein A reported that friction was a minor factor. Probably friction forces can be neglected in riveted joints but should be included in bolted joints, especially when clamping forces are high [7]. The rivet installation procedure has an effect on fatigue response. Whether a rivet is squeezed or bucked makes a difference to fatigue lives in a study of Szolwinski found that thin sheets, increasing the squeeze force has been shown to increase fatigue lives. It has less influence on thicker sheets [8]. Aluminum alloy 2014-T6 material is considered for all the structural elements of the panel. A two-dimensional finite element-analysis will be carried out on the splice joint panel. Distribution of fasteners loads and local stress field at rivet locations will be studied from finite element analysis [9]. A comparison will be made between the different methods of global FEM and subsequent stress analysis. This work also followed by a calculation of fatigue life to crack initiation at the highest stress location in the panel. Such calculations will be carried out for all different structural elements of the Riveted panel. Finally structural integrity evaluation is carried out through finite element analysis. After the experimental validation of strain field under static loading a fatigue testing will be carried out under the hoop stress of the fuselage to determine the fatigue crack initiation life. This will be compared with the theoretical prediction.

2. Calculation for length of rivet required

![Figure 1. Rivet dimensions](image)

The largest of the aircraft structural components, there are two types of metal aircraft fuselages: Full monocoque and semi-monocoque. The full monocoque fuselage has fewer internal parts and a more highly stressed skin than the semi-monocoque fuselage, which uses internal bracing to obtain its strength [10]. The 2000 series Al alloy (possessing magnesium, manganese, copper alloy) has high fracture toughness and resistance to crack propagation. It has good Mach inability and possesses good resistance.

3. Finite Element Analysis

Once after meshing and before proceeding further, we check for elements so that no elements fail during the application of the load. The elements which failed under the specific condition was checked and modified so that all the elements fell under the pre-defined standards and hence we ended up with a mesh where none of the elements failed [11].
The elements were checked for the following criteria.

**Table 1. Calculation for length of rivet required**

| Diameter of rivet | Volume of buckled head | Length required (without grip length) | Grip length (Considering the clearance allowed) |
|-------------------|------------------------|---------------------------------------|-----------------------------------------------|
|                   | Minimum                 | Maximum                               | Minimum                         | Maximum                   |
| d                 | A                       | B                                     | C                               |
| 2.4               | 7.92                    | 11.12                                 | 1.75                            | 2.46                      | (X+Y)X1.085 |
| 3.2               | 18.76                   | 26.35                                 | 2.33                            | 3.28                      | (X+Y)X1.063 |
| 4.0               | 36.64                   | 51.47                                 | 2.92                            | 4.10                      | (X+Y)X1.050 |
| 4.8               | 63.32                   | 88.95                                 | 3.50                            | 4.92                      | (X+Y)X1.096 |
| 5.0               | 71.57                   | 100.53                                | 3.65                            | 5.12                      | (X+Y)X1.082 |
| 6.0               | 123.68                  | 173.72                                | 4.37                            | 6.14                      | (X+Y)X1.067 |
| 6.4               | 143.17                  | 201.11                                | 4.59                            | 6.45                      | (X+Y)X1.064 |
| 8.0               | 293.16                  | 411.79                                | 5.83                            | 8.19                      | (X+Y)X1.076 |

A segment of the fuselage is considered for the finite element analysis. The splice joint of the fuselage is simulated in the model. Two semi-circular segments of the fuselage are modelled individually. The rivets in the joint are represented in the model using beam elements. The global stress response of the structure is captured through the fuselage segment analysis. This stress response is used as a benchmark reference for all iterative local analyses. The details of the stress analysis are presented in the following sections.
3.1. Geometrical specifications

![Figure 2. Fuselage Dimensions](image)

The following grid points and elements used for meshing of a fuselage structure are shown below.

- Number of grid points = 93264
- Number of CBAR elements = 1200
- Number of CQUAD4 elements = 88800
- Number of CTRIA3 elements = 3600

3.2. Load and boundary conditions

![Figure 3. Fuselage mesh with boundary condition and internal pressure](image)

The loading conditions that an aircraft could encounter under various conditions are many. In the current study only the internal fuselage pressure (also known as Cabin pressurization loading case) is considered as shown in fig 4. In the current project, the data is provided for the load case in which the fuselage is subjected to an internal pressure of 6.35psi. All six degrees of freedom are constrained at both end of the fuselage structure as shown in the figure 5. So when pressure applied in a fuselage it will be acting well within in the fuselage structure [12].
4. Results and Discussion

The stress contour obtained from the analysis is shown in figure 4. A maximum principal stress of 42.2 MPa can be observed from the stress contour. Maximum stress is observed at the rivet locations at the joint. The Y direction stress contour is also shown in figure 5.

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 from finite element analysis. This analysis ensures that the stress fields around the rivet holes are identical between the fuselage and the flat panel [13]. The flat panel is being designed to achieve this equivalence. Aluminum alloy 2014-T6 material is considered for all the structural elements of the panel [14]. Individual stress analysis is carried out on the panel and the results are discussed in the following sections.

The flat panel is considered from the selected area of a fuselage. The riveting standards are maintained while designing flat panel for analysis & testing. The two skins are riveted by using splicer as shown in fig 6. Stress contour from the fuselage analysis is considered as a reference for the local panel analysis.
Figure 6. Dimensions of flat panel

The figure 7 shows the boundary conditions of a riveted flat panel. The boundary conditions play a crucial role in capturing the true response of the structure. The predictions obtained from the analysis are compared with test results. In this case where X-direction is allowed for load to act at both ends, where tension-tension load acting at either end. At the mid of the panel near the joint a boundary condition with Z-direction displacement constraint is applied on the splice plate.

Figure 7. Boundary conditions of a riveted flat pane

The above fig.7 shows loading conditions of riveted panel. Load of 7000N is applied at both end of the riveted panel as just it will be acting as a tension-tension load. In the current project, the data is provided for the load case in which the fuselage is subjected to an internal pressure of 6.35 psi i.e., 0.04374 N/mm². This load is converted in to appropriate hoop stress value of 24.30 N/mm².

The following set of four figures shows the stress contours on the panel for the above described boundary condition and loading. A maximum principal stress of 40.3MPa can be observed from the stress contour 15]. In Fig 8 the X component, Y component and mid principal stresses are shown in the figures. Out of the entire stress contour the maximum principal stress is the one which shows the maximum stress value. The maximum stress is obtained at one of the rivet location [16].
From a stress analysis, Fig 9, it is clearly observed there is 4.5% variation between flat panel stress and stress location. Hence one can predict stress from flat panel instead of analyzing entire fuselage. Therefore experiment is carried out by using flat panel then it is co-related with fuselage. The above stress contour as shown in fig 10 the max principal stress at rivet location of fuselage structure and riveted panel, where riveted location plays a vital role in a stress analysis. We see almost similar stress contour of fuselage and panel.

**Figure 8.** Stress contour of flat panel, at X-Component

**Figure 9.** Stress (MPa) V/s No. of stress location
Figure 10. Stress variation along the width of the skin panel (outer row of rivet)

5. Conclusion
The allowable stress for the material under consideration is 350 N/mm$^2$ and from the analysis, it is found that the maximum induced stress in the structure is 42.2N/mm$^2$. This means that the structure is well within the safe limit for the load case considered. There is 4.5% variation between flat panel analysis and fuselage analysis in finite element analysis. As the maximum stress location is found at the rivet region. There should be detailed inspection of the structure at this critical locations i.e. rivet region. The prediction method of fatigue life helps the maintenance personnel by giving smaller number of inspections. Investigating and improving the riveting process by prediction methods of fatigue life. To increase the fatigue life of riveted joints designer should take care of riveted design and riveting process. Manufacturing aspects plays a vital role in a fatigue life of an aircraft. Even minor default in a riveting process there will be crack initiated at the structure within the predicted fatigue life.

References

[1] Huth H. 1986 Fatigue in mechanically fastened Composite and metallic joints ASTM STP 927
[2] Newman JC, Harris CE, James MA, Shiva Kumar K.N 1997 Fatigue in aging aircraft 523–52
[3] Hartmann EC, Marshall H, Eaton ID 1954 NACA-TN3269
[4] Gopinath KV 1992 American Institute of Aeronautics and Astronautics 92
[5] K. P. Ashik and Ramesh S. Sharma 2015 Advanced composite Materials 21 1568–1577
[6] Ramanan G and Edwin Raja Dhas J 2017 Int. J. of Mech. Eng. Tech 08 866
[7] Yuchen Wen, Xiaowei Yue, Jeffrey H. Hunt, Jianjun Shi 2018 J of Manuf Systems 46 272–281
[8] Maher Bouazizi, Tarek Lazghab, Mohamed Soula 2018 Thin-Walled Structures 127 40–50
[9] Skobodanka Boljanović, Stevan Maksimović 2017 Procedia Structural Integrity 5 801–808
[10] Sharaschandra Kusugai, R. S. Kadadearvarmath D 2017 Materials Today: Proc 4 10696–703
[11] Jinsan Ju, Xiaochuan 2013 Mathematical and Computer Modelling 58 627–633
[12] Ramanan G and Edwin Raja Dhas J 2017 Int. J. of vehicle structures and sys 5 309
[13] M. Quatmann, N. Aswini, H. G. Reimerdes, N. K. Gupta 2013 Aero Sci and Tech 27 76–83
[14] S. M. O. Tavares, P. M. S. T. de Castro 2011 Engineering Fracture Mechanics 78 2907–2918
[15] Lucjan Witek 2006 Engineering Failure Analysis 13 572–581
[16] M. Skorupa, T. Machniewicz, A. Skorupa, A. Korbel 2015 Int. Journal of Fatigue 80 417–425