Structural analysis of Aircraft fuselage splice joint

R Udaya Prakash 1, G Raj Kumar 2, R Vijayanandh 3, M Senthil Kumar 4 and T Ramganesh 5

1 - B, E student, 2, 3 - Assistant Professor, 4 - Assistant Professor (SRG), 5 - Project Engineer
Department of Aeronautical engineering, Kumaraguru College of Technology, Coimbatore - 641049, Tamil Nadu, India

Email: vijayanandh.raja@gmail.com

Abstract. In Aviation sector, composite materials and its application to each component are one of the prime factors of consideration due to the high strength to weight ratio, design flexibility and non-corrosive so that the composite materials are widely used in the low weight constructions and also it can be treated as a suitable alternative to metals. The objective of this paper is to estimate and compare the suitability of a composite skin joint in an aircraft fuselage with different joints by simulating the displacement, normal stress, vonmises stress and shear stress with the help of numerical solution methods. The reference Z-stringer component of this paper is modeled by CATIA and numerical simulation is carried out by ANSYS has been used for splice joint presents in the aircraft fuselage with three combinations of joints such as riveted joint, bonded joint and hybrid joint. Nowadays the stringers are using to avoid buckling of fuselage skin, it has joined together by rivets and they are connected end to end by splice joint. Design and static analysis of three-dimensional models of joints such as bonded, riveted and hybrid are carried out and results are compared.

1. Introduction
Composite materials are broadly using in the different industries such as automotive, chemical, electrical industry, aerospace and low-temperature application based organization. Composites have been one of the materials used for repairing the obtainable structures in different applications due to the reason of its advanced methodologies and properties, still now the joining process in composite parts are handling by adhesives or mechanical joints. In this paper, a new method called hybrid joint have been introduced in the joining process of composite material, the hybrid joint is nothing but a combination of both adhesive and mechanical joint. An attempt has been made to analyze the stress distribution in 3D models of three configurations of splice joint in stringers namely bonded, riveted, and hybrid. A major advantage of adhesive bonds with riveted joint is stronger than the ultimate strength of many metals so it can be proposed in the fuselage skin joints in aircraft structures.

2. Problem Analysis
2.1 Related works
Adarsh et al. (2012) have studied about the stress analysis and fatigue life prediction for finite element analysis of a splice joint in an aircraft fuselage. The major load boundary condition for this problem is cabin pressurization, and the innovative is splice joint implementation in fuselage structural members. A typical splice joint panel consisting of skin plates, doubler plate, and a longitudinal stiffener is considered for this study [1]. The global finite element analysis of a typical fuselage segment been carried out and the response of the splice joint was evaluated. The splice joint is one of the dangerous locations for fatigue crack to initiate; hence the prediction of fatigue life for crack initiation been carried
out at maximum stress location, this splice joint based connection in aircraft for more lifetime of components was proposed by Jaap (2009) which are deals with aircraft structural design, load spectra and production techniques [2]. Xiong and Bedair (1999) mentioned about modeling steps for the stress investigation of riveted lap joints in aircraft structure. Analytical methods have been developed depending upon the difficult variation approach for lap joints with single or multiple rivet holes, in this work the connection members can be either metallic or composite materials. The stresses in the two joined plates and the rivet loads are determined [3]. Amarendra (2006) conducted a study where the main objective of the research was to establish a link between critical riveting process parameters on the strength of fatigue damage in the joint. Aircraft fuselage splices are fatigue dangerous structures and the damage coupled with these structures has been widely accepted as a safety issue that needs to be addressed in the structural integrity of aging aircraft [4]. Floyd (2010) determined the applicability of a technique to forecast the number of cycles of fatigue loading of a structure to failure [5]. Karthick N et al. tells about the current study includes a panel, which represents the fuselage splice joint, that means splice joint is a location where it experiences the resisting force region at number of rivet locations in a single row. Fatigue cracks will emanate from the rivet holes, which was modified as uniform stress due to the internal pressure [6]. A. Rukesh Reddy et al. investigate about the maximum stress concentration part of the splice joint of an aircraft wing bottom surface due to tensile loading. Splicing is generally used to retain a clean aerodynamic surface of the surface for most of the aircraft structure. The structural analysis of the joint is carried out to compute the structural parameters such as displacement, stress, etc. at rivet holes due to bearing and bending load. With the effect of the new modification in joints, the rivet hole local stress is optimized to minimize [7]. Vijayaraja L et al. Presents work the correlation of Z-Stringer splice joint present in Aircraft fuselage skin with three combinations, they are metallic, composite metallic and composite; the stringer was assembled in fuselage skin by different fasteners and has been connected end to end by splice joint [8]. Basil sunny et al. the aim of this paper is to compare different splice joint properties in an aircraft fuselage with the prediction of fatigue life to crack ignition using the composite material. The application of the composite material in the aircraft is a chance to increase the strength to weight ratio. By replacing aluminum alloys by composite material i.e., Kevlar 49 the fatigue life of the fuselage can be increased [9].

2.2 Problem solution technique
The major challenges in today’s aircraft industry are to overcome the increasing demands of lower weight, longer life of components, higher stability and performance, all this at a reasonable cost and in a limited period of time. The fuselage structure generally consists of skin panels joined directly to the structural members such as frames, stringers for longitudinal splices. In assembling process critical structures like military or commercial aircraft, riveted or bolted joints are basically used as they offer many options to the engineer. To fulfill fatigue conditions, the engineer can either keep the resisting force levels below the survival limit or make sure that the slow fracture growth life of the component is larger than the estimated value plus designed factor of safety. The failure reason of aircraft components was mainly initiated by external forces such as fuselage pressurization force, wing lifting force. The force due to the fuselage cabin pressurization can be considered as one of the essential loads cases for the fuselage structure damage which experiences constant amplitude load cycles; similarly, the force due to the wing surface pressure difference can be included as one of the major reason for wing damage [10, 11]. To overcome these challenges, splice joints are used for the fuselage and wing structures. A splice joint is a method of joining two members from one end to other with the help of another structural member. It is an alternative to other joints such as the butt joint and the lap joint. Splice joints are stronger than unreinforced butt joints and have the potential to be stronger than a lap joint. They are more visible than a lap joint but may be preferred when more strength is required. Splices are therefore most often used when structural members are required in larger lengths than the available member. The general form of the splice joint is the half lap splice member, which is used in building construction, where it is common to join smaller lengths of timber into longer beams. A typical splice joint panel consisting of skin plate, extruded plate, and a longitudinal stiffener cabin pressure results in the radial
growth of the skin and this radial growth is resisted by rings and stringers giving local bending along the fastener lines [12, 13].

3. Preliminary details

3.1 Design of different Aircraft joints

The design is the creation of convention for the development of an object. The following steps are will be considered with a good explanation that is recognition of need, problem definition, synthesis, analysis and optimization, evaluation by expert and presentation or implementation [14]. In this, paper the design of aircraft fuselage joints modeled by CATIA. Table 1 shows the details about parameters involved in the analysis. The adhesive layer between the laminates of thickness 0.1mm and aluminum rivet are used; the models of bonded, riveted, and hybrid joints are shown in figure 1-3.

| Sl No | Parameters               | Value (mm) |
|-------|--------------------------|------------|
| 01    | Stringer height (hstr)   | 35         |
| 02    | Stringer width (dstr)    | 20         |
| 03    | Stringer length          | 100        |
| 04    | Stringer thickness       | 1          |
| 05    | Splice plate height (hsp) | 30        |
| 06    | Splice plate length (lsp) | 60        |
| 07    | Splice plate thickness   | 1          |
| 08    | Diameter of the holes in plate | 5 |
| 09    | Diameter of the rivet head | 5        |
| 10    | Height of the rivet head  | 5          |

3.2 Discretization

Discretization is the process of dividing the test object into a finite number of small blocks namely called as an element, which is associated with nodes. In finite element analysis approach, the results are only predictable at the nodes, so the number of nodes increases in order to increase the accuracy of a given model. Discretization processes involved in these models are mostly divided by hexagonal elements because of its critical design and dimension and to implement the natural effects in the result in order to increase the result the adherent, adhesive were glued together and finer mesh was used in the design [15]. The meshed models of joints such as bonded, riveted and hybrid are shown in figure 4-6 and table 2 shows the comparison of a number of nodes and number of elements for riveted, bonded and hybrid joints.
Table 2. Meshing comparison

| Joints   | Number of elements | Number of nodes |
|----------|--------------------|-----------------|
| Riveted  | 141566             | 260923          |
| Bonded   | 77503              | 154346          |
| Hybrid   | 212261             | 402292          |

Figure 4. Meshed model of Bonded joint  
Figure 5. Meshed model of Riveted joint  
Figure 6. Meshed model of Hybrid joint

4. Result and Discussions

4.1 Selection of Material process

In structural analysis, selection of material process and its mechanical properties like Young’s Modulus, Poisson ratio, etc. of an object plays a major role. Carbon fiber composite is used for modeling and static analysis of 3D models of joints (bonded, riveted, hybrid) was carried out using ANSYS. The normal stress contours, shear stress contours, displacement over the joints and vonmises contours were plotted. The primary objective of this study is to examine the cause of different parameters, i.e. adherents, adhesive thickness and overlap length, on the crash force and breakdown mode of joints with disparate materials. Therefore, for this important purpose mild steel and carbon fiber reinforced polymer laminates have been included as the structural adherent materials. The multi-disciplinary design of aircraft structural design has been considered for gathering geometric dimensions for modeling and mechanical loads acting on the joint and the specimens are tested under a uni-axial tensile quasi-static displacement. The load-deformation, load-strains readings is noticed and conclusions concerning the joints stiffness, strength are obtained [16]. The modeled static analysis of strain and stress distribution over joints of stringer splice elements is made from the carbon-fiber and from the 2024 aluminum alloy. Table 3 shows the comparison of different materials properties.

Table 3. Material properties

| Type of material | Young’s Modulus (GPa) | Density (Kg/mm$^3$) | Poisson ratio |
|------------------|-----------------------|---------------------|---------------|
| Carbon fiber     | 150                   | 1.6x10$^{-6}$       | 0.26          |
| Aluminum         | 70                    | 2.71x10$^{-6}$      | 0.33          |
| Epoxy            | 4                     | 1.08x10$^{-6}$      | 0.4           |

Aluminum 2024 material is used for rivets, the carbon fiber material is used for analysis, and stress values are evaluated. One end of the model was constrained from x, y, z translations, and the other end the load of 10 MN is applied to this end towards the positive x-direction.
4.2 Static Structural Analysis of Joints
A static analysis is the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. The static analysis determines the displacements, strains, stresses, and forces in structural members or components caused by loads unable to induce significant inertia and damping effects. Steady loading process and response behaviors are assumed; that is, the loads and the retort of structural components are assumed to vary slowly with respect to time. The basic methods involved in the structural analysis are flexibility and stiffness methods. The structural analysis of aircraft component splice joint is simulated with the help of numerical algorithm in order to determine the response of a structure to specified loads and actions. This response behavior is measured by determining the internal forces and displacements throughout the structure [17].

4.2.1 Bonded joints
The contour images of the bonded joints analysis result of vonmises, shear stress, normal stress and total deformation shown in figure 7-10. The maximum value of vonmises stress is 910.04 N/mm². The maximum value of shear stress is 140.43 N/mm². The maximum value of normal stress in bonded joint was found to be 479.01 N/mm². The maximum value of total deformation is found to be 0.3990 mm; it was located on the two plates bonded portion.

![Figure 7. Vonmises stress of bonded joint](image)
![Figure 8. Normal stress of bonded joint](image)

![Figure 9. Shear stress of bonded joint](image)
![Figure 10. Total deformation of bonded joint](image)

4.2.2 Riveted joints
The contour plots of the riveted joints analysis result of normal stress, vonmises, shear stress, and total deformation shown in figure 11-14. The utmost rate of vonmises stress for riveted joint is 1330.9 N/mm². The maximum value of shear stress is 267.71 N/mm². The rivet is meshed by using 8-node brick element and the composite laminates are designed and meshed using hexagonal element. The maximum value of
normal stress is found to be $292.58 \text{ N/mm}^2$ and the maximum value of total deformation was found to be 0.2242 mm, it was located at the head of the rivet portion.

4.2.3 Hybrid joints
The design of this joint is similar to the riveted joint except that it has a thin layer of adhesive between the laminates. The utmost amount of vonmises stress is found to be $374.01 \text{ N/mm}^2$. The maximum value of shear stress was found to be $103.72 \text{ N/mm}^2$. The maximum value of normal stress is found to be $280.75 \text{ N/mm}^2$. The total deformation is 0.1384 mm around the riveted region. The images of the analysis result of vonmises, shear stress, normal stress and total deformation of the hybrid joint are shown in the figure 15-18.
The figures 7-18 shows the comparison of stress contours between bonded, riveted and hybrid joints; it clearly explains how it was distributed throughout the laminate and the adhesive took up much of the load in hybrid joint, since applying load acts on the joint. Whereas in the riveted joint, the rivet head, and the laminate plane take up more resisting force compared to the inner of the laminate and the rivet shank [18], in the case of the hybrid joint is found that the stress amounts, total deformation is less when compared to bonded, riveted joints. Hence, the hybrid joint is best suitable alternate joint for critical working environments and the values of normal stress, vonmises stress, shear stress, and total deformation is tabulated and compared below. Table 4, figure 19 and 20 shows the comparison stresses and deformation results for different joints.

Table 4. Comparison of stress results

| Types of joints | Vonmises stress (N/mm²) | Shear stress (N/mm²) | Normal stress (N/mm²) | Joint Deformation (mm) |
|-----------------|-------------------------|---------------------|-----------------------|------------------------|
| Bonded          | 910.04                  | 140.43              | 479.01                | 0.3990                 |
| Riveted         | 1330.9                  | 267.71              | 292.58                | 0.2242                 |
| Hybrid          | 374.01                  | 103.72              | 280.75                | 0.1384                 |

Conclusion

In this analysis, the simulation of stress distribution and deformation in joints such as bonded, riveted and hybrid joints has been carried out successfully. The stress and deformation values are used to compare the results with three joining methods. The equivalent stress for a hybrid joint is found to be less compared to rivet...
and bonded joint. The stress induced by using ANSYS is less than the materials ultimate stress and ultimate limit. The total deformation for a hybrid joint is comparatively less than other two joints. It is found that a well-designed hybrid joint is very efficient and preferable when compared to bonded, riveted joints as they cause least deformation and efficient stress distribution.

References
[1] Design and Analysis of Hybrid Composite Lap Joint Using Finite Element Methods, S.Venkateswarlu, K.Rajasekhar, International Journal of Engineering Trends and Technology (IJETT), sep 2013.
[2] W.Hufenbach a, L.A.Dobrzaniś, b, M.Gude a, J.Konieczny b, A.Czulak a, “Optimization of the rivet joints of the CFRP composite material and aluminum alloy” Achievements in Materials and Manufacturing Engineering Volume 20 (2007).
[3] The multi-disciplinary design of aircraft structural design michel van tooren and lars krakers, delft university of technology, netherland.
[4] “Stress analysis of a splice joint in an aircraft fuselage with the prediction of fatigue life to crack initiation” by basil sunny, richu thomas, mechanical department, amaljyothi college of Engg. Kanjirappally, India, an international journal of design and manufacturing technology (ijdmt), issn 0976 – 6995 (print), issn 0976 – 7002 (online), volume 5, issue 3, September - December (2014), pp. 142-147.
[5] “Simulation and design of splice joint in fuselage” by vijayaraja, a r anwar khan, department of mechanical engineering ghousia college of engineering, ramanagaram, vtu-belagavi, karnataka, India, published by ijret: international journal of research in engineering and technology eissn: 2319-1163 | pssn: 2321-7308, volume: 04 issue: 07 | july-2015.
[6] “Analysis of the Fuselage Structure for Multi-Site Damage” by Karthik N, Dr. C Anil Kumar, Department of Mechanical Engineering, KSIT, Bangalore, Karnataka, India published by International Journal of Innovative Research in Science, Engineering, and Technology, ISSN: 2319-8753 Vol. 2, Issue 7, July 2013.
[7] ‘Stress Analysis of Splice Joint of the Aircraft Bottom Wing Skin by Finite Element Method’, A.Rukesh Reddy.P. Ramesh, .B. Siddeswara rao, Department of Mechanical Engineering, Sri Venkateswara college of engineering and technology,Chittoor- 517002 published by International Journal of Computational Engineering Research, Vol. 03, Issue, 11,Page 13 - 17, Issn 2250-3005, November 2013.
[8] “Stress Analyses of Mechanically Fastened Joints in Aircraft Fuselages “by J.J.M. de Rijck, S.A. Fawaz, J. Schijve, R. Benedictus and J.J. Homan published in 24th ICAF Symposium - Naples, 16-18 May 2007.
[9] “Stress Analysis and Fatigue Life Prediction for Splice Joint in an Aircraft Fuselage through an FEM Approach” by Adarsh Adeppa, Dr. M. S. Patil, K. E. Girish published in International Journal of Engineering and Innovative Technology (IJEIT), Volume 1, Issue 4, April 2012, ISSN: 2277-3754.
[10] Fawaz, S. A., and BörjeAndersson. “Accurate Stress Intensity Factor Solutions for UnsymmetricCornerCracks at a Hole”. Proc. of the Fourth Joint NASA Conference on Aging Aircraft, vol 15 (2000),pp 135-139
[11] C.S. Kusko, J.N. Dupont, A.R. Marder, “Influence of stress ratio on fatigue crack propagation behavior of stainless steel welds”. Welding Journal, vol 19,(2004), pp 122-130,
[12] N. Ranganathan, H. Aldroe, F. Lacroix, F. Chalon, R. Leroy, A. Tougui. “Fatigue crack initiation at a notch”. International Journal of Fatigue, vol 33, (2011), pp 492–499.
[13] Lance Proctor et al, local analysis of fastener holes using the linear gap technology using MSC/NASTRAN, Presented at MSC Aerospace Users’ Conference, 2000, pp-1-24.
[14] A.M.Brown, Simulating fretting contact in single lap splices, International Journal of Fatigue, 2009, pp 375-384.
[15] M.R. Urban, Analysis of the fatigue life of riveted sheet metal helicopter airframe joints, International journal of fatigue, 2003, pp 1013-1026.
[16] Gresnigt AM, Steenhuis CM. The stiffness of lap joints with preloaded bolts, Proceedings of the NATO ARW, 2000.
[17] Brombolich LJ. Elastic–plastic analysis of stresses near fastener holes. AIAA 11th Aerospace Sciences Meeting, 1973. [AIAA no. 72-252].
[18] S.G.S. Raman, V.M. Radhakrishnan, “On cyclic stress-strain behavior and low cycle fatigue life,” Materials and Design, 2002, Vol.23, pp.249- 254.