Circumferential Joint Analysis in Aircraft Structure

Shivakumar Chetty[1] Sharanabasappa Bhurke[2]
[1]PG Scholar [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]Don Bosco Institute of Technology, Bengaluru, India

Abstract— The stringers and frames are used in aircraft structure or fuselage to avoid buckling. Stringers and frames are assembled in fuselage using bolts and are connected end to end by means of splice joints. In this paper, Static and dynamic analysis on circumferential joints of an aircraft structure with L joint and U joint for both linear and Bilinear analysis using Aluminium alloy is carried out. At present 3D modeling of L type splice joint and U type splice joints are carried out in UNIGRAPHICS and is analysed using FEM approach in ANSYS for both static and dynamic load conditions. Also validation is carried out with analytical and FEM methods.

Key words: Aircraft, Fuselage, Stringers, Frames, Splice joint

I. Introduction

Aircraft structure happens to be one of the most sensitive structures which consist of frames, skin panels and strings which are assembled through clips and ducts. The joints are very sensitive because they are fastened with rivets and splice. The frame of a fuselage is either made of composite or metallic depending on the load path. The skin panel is however composite. The strings are linear structure which provides axial stiffness to fuselage. Splice joints are used for the fuselage structure.[1]

Stringer is a little thin portion of material which is attached to the skin of the air ship. In the fuselage, stringers are connected to edges which are in the longitudinal heading of the aircraft. The biggest single thing of the fuselage structure is the skin and its stringer. It is additionally the most discriminating structure since it conveys the greater part of the essential loads because of fuselage twisting, shear, torsion and internal compression. End to end of stringers is connected by splice joint [2].The investigations are carried out on splice joints for Aircraft structure. The circular frames provide circumferential stiffness to the structure without allowing the structure to buckle. Design and simulation of these circular frames is very challenging. The splice joints play a vital role in providing structural integrity in framed structure. [3] Design of splice joint to take up circumferential hoop loads and axial loads is a challenge. Design of bolts/fastens and the festering fusion becomes important since the behaviour of bolts with pretensions in the structure for the critical load path is very difficult to predict. Hence, it calls for sensitive analysis. [4]

II. Objectives

Estimation of linear stress, strain and deformation of a high longitudinal load transfer (U splice) circumferential joint by linear static structural analysis.

Estimation of linear stress, strain and deformation of a low longitudinal load transfer (L splice joint) circumferential joint by linear static structural analysis.

Estimation of Bi-linear stress, strain and deformation of a high longitudinal load transfer (U splice) circumferential joint by linear static structural analysis.

Estimation of Bi-linear stress, strain and deformation of a low longitudinal load transfer (L splice joint) circumferential joint by linear static structural analysis.

Dynamic analysis to find the different mode of circumferential joint analysis of Aircraft structure.

III. Methodology

A. Geometric Modeling

The first step in project work is to select the product to be developed or improved. During the development process a 3D model is created.

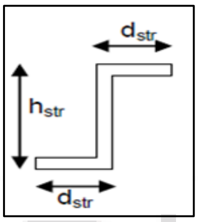


Fig. 1: The section of stringer

Sl.no	Name	Dimensions
1	Stringer height(hstr)	35mm
2	Stringer width(dstr)	20mm
3	Stringer length	80mm

Table 1: Dimensions as above

The advantage of making 3D model is that it allows fast variations in the model geometry. For this purpose ANSYS or UNIGRAPHICS will be used. Small fillets and blends will be ignored while creating the model .Here the shapes are studied and are mostly two- or three-dimensional.

Material	Young's modulus	Yield strength	Ultimate strength	Poisson's ratio
AL 2024 T35I	73 MPa	280 MPa	470 MPa	0.33

Table 2: Splice is 30mm wide and 60mm long, Bolt dia 10mm

B. Mathematical Modelling

During the process of work the geometry of the 3D CAD model created using ANSYS or UG is imported in the Finite Elemental Analysis (FEA) software for various tests. The various tests include stress, strain displacements.

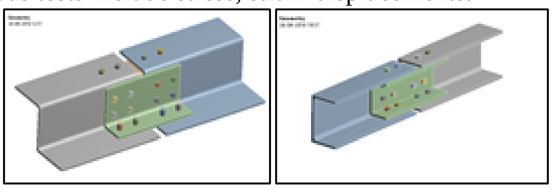
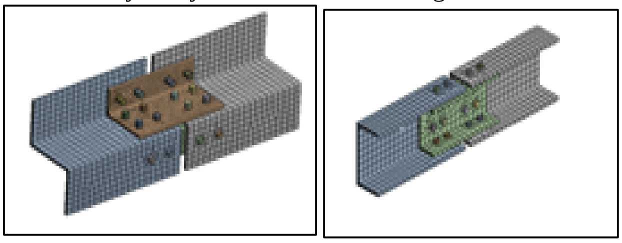


Fig. 2: Isometric view of L-type and U- type splice joint

C. Finite Element Model:

After this step the model is meshed using automatic mesh generator in ANSYS Workbench. After Mesh Generation boundary conditions are defined and are applied to the model which is then analysed using ANSYS. Then the results are post processed using ANSYS Workbench and will be clearly analysed for the better design.



Nodes 25075, Elements 5364 Nodes 12362, Elements 2645
Fig. 3: Meshed model of L- type and U- type splice joint

IV. Finite Element Approach

Finite element Method based upon discritization of component into Finite number of blocks (elements),Finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It uses subdivision of a whole problem domain into simpler parts, called finite elements, and variational methods from the calculus of variations to solve the problem by minimizing an associated error function.

Analogous to the idea that connecting many tiny straight lines can approximate a larger circle, FEM encompasses methods for connecting many simple element equations over many small subdomains, named finite elements, to approximate a more complex equation over a larger domain.

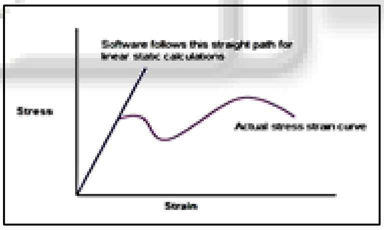


Fig. 4: Stress-strain curve

A. Linear Static Analysis

It is the simplest and most commonly used type of analysis.

Linear means straight line. σ = €E is the equation of straight line (y=mx+c) passing through origin. 'E', Young's modulus is the slope of curve and is a constant. In real life, after passing yield point material follows nonlinear curve but software follows same straight line. Component break into two separate pieces after crossing ultimate stress but software based analysis never show failure in this fashion.

It shows single unbroken part only with red color zone at the location of failure. Analyst has to conclude whether the component is safe or fail by comparing maximum stress value with the yield or ultimate stress.

There are two conditions for static analysis:

No variation of force with respect to time (Dead weight)

Equilibrium condition - ∑ Force = 0 and ∑Moments = 0.

Hence, peak linear stress linear strain will be calculated by this method for dead frontal axle under defined loading conditions.

V. Validation

A. Analytical Solution

Axial load F= 10 KN = 10^4 N
Thickness of plate h= 10 mm
Width of plate W= 50 mm
Diameter of hole d or a= 20 mm
For two hole d/w = 20/50
From DDHB for d/w = 0.4
Theoretical stress concentration factor k_σ= 2.2
We have,
σ_{nom}= F/(w-2d)h =10000/(50-2×10)10 = 33.33 N/mm^2
K_σ = (σ_{max})/σ_{nom} =   2.2 = σmax/33.33
Maximum stresses σ_{max}= 73.326 N/mm^2
Shear stress for double bolt
Shear stress = 2F/πd^2 = (2×10000)/(π×10^2)
= 20000/314.1592
Shear stress = 63.66 N/mm^2

B. FEM Analysis

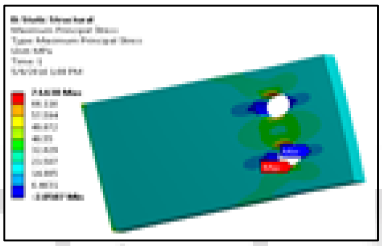


Fig 5: Max principal stresses

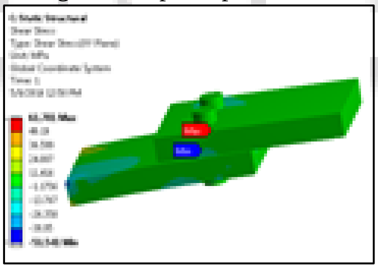
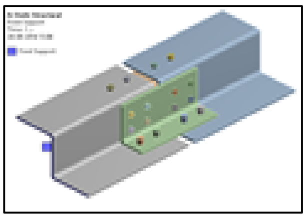


Fig. 6: Shear stresses

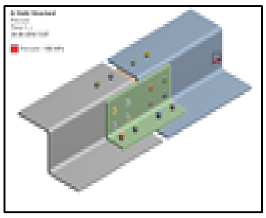
Particulars	Results	Analytical solution	FEM solution	% Error
Plate	Max principal stresses (N/mm2)	73.326	74.638	1.75%
Bolt	Shear stresses (N/mm2)	63.66	61.78	2.95%

Table 3: Comparison of results

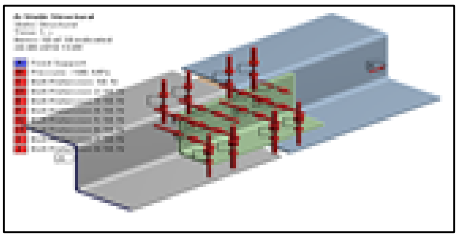
VI. Boundary Conditions



Fixed support



Applied pressure

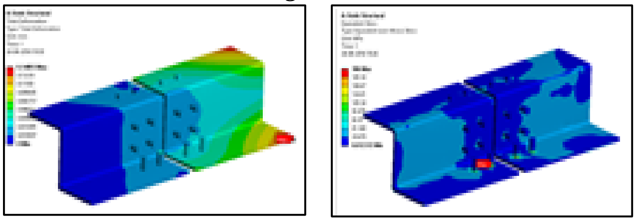


Bolt pretension to all bolts

Fig. 7: Different boundary conditions applied

VII. RESULTS & DISCUSSION

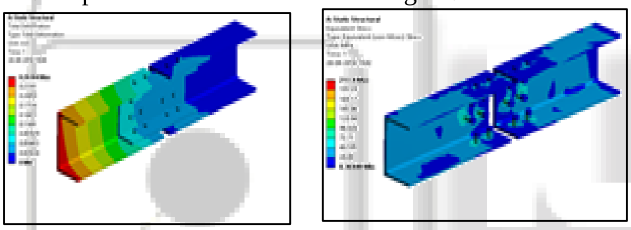
The Total deformation obtained by FEM analysis is 0.14893 mm and Equivalent von- mises stress obtained is 186 Mpa and is as shown in below figure.



Total deformation Von- mises stresses

Fig. 8: Linear static structural analysis (L-type)

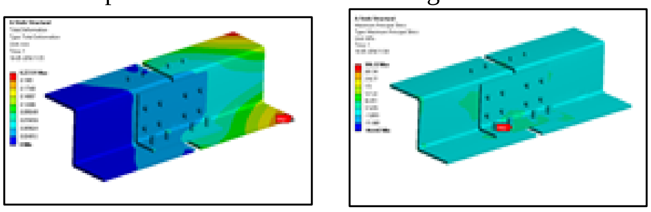
The Total deformation obtained by FEM analysis is 0.26306 mm and Equivalent von- mises stress obtained is 217.4 Mpa and is as shown in below figure.



Total deformation Von- mises stresses

Fig. 9: Linear static structural analysis (U- type)

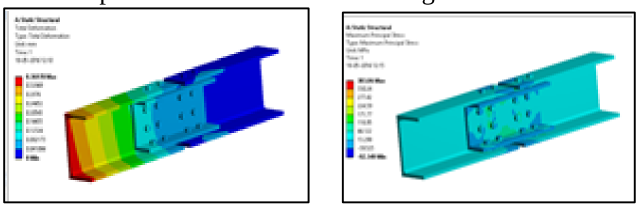
The Total deformation obtained by FEM analysis is 0.22331 mm and Maximum principal stress obtained is 306.32 Mpa and is as shown in below figure.



Total deformation Maximum principal stresses

Fig. 10: Bi-Linear static structural analysis (L-type)

The Total deformation obtained by FEM analysis is 0.36978 mm and Maximum principal stress obtained is 383.06 Mpa and is as shown in below figure.

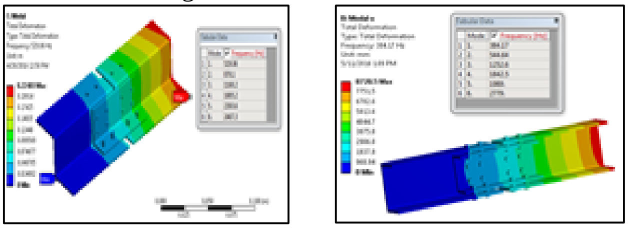


Total deformation Maximum principal stresses

Fig 11: Bi-Linear static structural analysis (U-type)

Natural frequency at mode 1 obtained is 529.06 Hz to mode 6 of frequency 2467.3 Hz for L- joint.

Similarly, Natural frequency at mode 1 is 384.17 Hz to mode 6 of frequency 2779 Hz for U- joint and is as shown in below figure.



Modal analysis L- type Modal analysis U- type

Fig. 10: Modal analysis for six different mode

VIII. CONCLUSION

From the results obtained, to conclude that

Maximum equivalent stress induced in L-type splice joint for linear static structural analysis is 186 MPa. Similarly, for U- type splice joint maximum equivalent stress is 217 MPa. Deformation by L-type is a 0.14893 which is less than 0.26304 for U- type joint.

The value obtained for L-type 186 MPa which is less than yield stress 280 MPa for AL-alloy material. By contrast, L-type splice joint is more preferable than a U- type joint to the fuselage of a circumferential joint of an aircraft structure for linear static structural analysis.

Maximum principal stress induced in L-type splice joint for linear static structural analysis is 306.32 MPa. Similarly, for U- type splice joint maximum principal stress is 383.06 MPa.

Deformation by L-type is a 0.22331 which is less than 0.36978 for U- type joint.

The value obtained for L-type 306.32 MPa, which is less than 383.06 MPa for U- type joint of AL-alloy material. By contrast, L-type splice joint is more preferable than a U- type joint to the fuselage of a circumferential joint of an aircraft structure for two- linear static structural analysis.

Vibration analysis is carried out to find a critical six primitive modes and corresponding natural frequency for L-type and U-type circumferential splice joint for both linear and bi-linear analysis of static and dynamic load conditions.

The results of maximum principal stress and shear stress obtained by both FEM method and analytical method are validated and acquired a percentage error of only 1.75%. Therefore, FEM validated through an analytical method.

ACKNOWLEDGEMENT

I am grateful to Mr. Sharanabasappa Asst. Professor, Dr.Rathnakar.G Professor and HOD ME and Dr.V.S. Ramamurthy Principal, Don Bosco Institute of Technology, Bengaluru for their cheerful encouragement and timely inspiration which helped me to complete the paper.

REFERENCES

[1] MichAel van Toren and Lars Krakaers Multi-disciplinary Design of Aircraft Fuselage Structures, Delft University of Technology, Netherland.

[2] Michel Chuin-Yunge Niu, Airframe Structural Design. California U.S.A march 1988.

[3] E.F Bruihn, B.S, M.S.C.E, Dr. Engg, Analysis and design of flight Vehicle Structures, 3rd Edition, June 1973.
[4] Stress Analysis of Splice Joint of the Aircraft Bottom Wing Skin by Finite Element Method. A. Rakesh Reddy, P.Ramesh,B.Siddeswara