Ramesha et al. 2014. *Int. J. Vehicle Structures & Systems*, 6(4), 100-103
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.6.4.03


**International Journal of Vehicle Structures & Systems**
Available online at www.maftree.org/eja

# Analysis of Axial Crack in Aircraft Fuselage Circumferential Joint using Finite Element Simulation

**H. Ramesha[a,b], N.C. Mahendra Babu[d] and H.V. Lakshminarayana[a,c]**

[a]*Dept. of Mech. Engg., Dayananda Sagar College of Engg., Bengaluru, India*
[b]*Corresponding Author, Email: bharatavarsh@gmail.com*
[c]*Email: hvl_mech2007@rediffmail.com*
[d]*Dept. of Mech. Engg., B.N.M. Institute of Technology, Bengaluru, India*
*Email: ncmbabu@gmail.com*

**ABSTRACT:**

*Ring frames, stringers and skin panels are the main parts of aircraft fuselage structure. Circumferential joint is formed by joining skin panels in circumferential direction using doubler plates and adhesive bonding. In this paper finite element (FE) modeling of circumferential butt joint of fuselage structure using commercial FEA software (ANSYS) is presented. The FE model is validated using a benchmark and then fracture mechanics results are graphically presented and discussed.*

**KEYWORDS:**

*Circumferential joint; Crack; Finite element modelling; Aircraft fuselage structure; Structural integrity*

**CITATION:**

H. Ramesha, N.C. Mahendra Babu and H.V. Lakshminarayana. 2014. Analysis of Axial Crack in Aircraft Fuselage Circumferential Joint using Finite Element Simulation, *Int. J. Vehicle Structures & Systems*, 6(4), 100-103. doi:10.4273/ijvss.6.4.03.

## 1. Introduction

Fatigue and fracture of fuselage structure is a major problem in aircraft design. Fracture mechanics provides a methodology for prediction, prevention and control of fracture and fatigue crack growth in materials, components and structures. Finite element (FE) modeling for computational fracture mechanics is therefore essential [1]. Ring frames stringers and skin panels are parts of aircraft fuselage structure and are designed to take aircraft loads [2]. Joint regions of skin panels are found critical for fatigue failure. Hence, detailed analysis for damage tolerant design is essential [3]. In this work an effort is made to analyze circumferential butt joint region of fuselage structure. Fracture mechanics analysis objectives are - (1) Stress analysis of cracks to derive crack tip stress field equations and define stress intensity factors, (2) Determination of crack tip stress intensity factors, (3) Predict mixed mode fracture under static, dynamic and sustained loads, (4) Prediction of fatigue crack growth under constant amplitude, variable amplitude and spectrum loads, (5) Prediction of residual strength and (6) Prediction of remaining life and NDI intervals.

FE modeling is defined here as the analyst's choice of material models (constitutive equations and failure criteria), elements, mesh, constraint equations, analysis procedures, governing matrix equations and their solution methods, specific pre and post processing options available in chosen FEA software for computational fracture mechanics. In this study the FE models are created using ANSYS software because the pre processing command KCON's enables a refined mesh of singular iso-parametric triangular shell elements to be created at each crack tip and a compatible mesh of regular elements triangular or quadrilateral in shape could be used. The post processing command KCAL enables computation of mixed mode stress intensity factors. The use of sub modeling capability enormously improves the computational efficiency. The graphical post processing capability enables critical regions to be identified where lead cracks are introduced. A significant enhancement in computation of mixed mode membrane and bending stress intensity factors is achieved using a special purpose post processing subprogram called 3MBSIF. Higher order element SHELL 281 is used for creation of FE model. This element contains 8 nodes with 6 degrees of freedom at each node. The FE model is validated using experimental results reported in [4].

## 2. FE modelling

Fig. 1 shows the specific joint region considered in this study. The structural panel shown in Fig. 2 includes 6 ring frames, 7 stringers and the fuselage skin. Table 1 shows a list of components, their thickness and material property data. FE modelling and validation of its results with available experimental data are the main objectives of this work. FE model is prepared as per the dimension details obtained from literature [4]. ANSYS modeling capability commands are used for this purpose. Minor geometrical details such as fillets, rivet holes and chamfers are excluded for the ease of analysis. From the solid model created, mid-surfaces of the components are extracted and are moved to a common plane where joints are required. Table 2 gives a list of joint regions in the model. FE model is created so as to get nodes at specific joint region and joint is simulated by merging corresponding nodes of the components. A typical assembly of the skin, stringer and a frame involves adhesive bonding. Fig. 3 presents the FE model of the panel and circumferential joint. The complete model was created using ANSYS pre processing capability. The mesh involves 143392 shell elements and 435124 nodes.

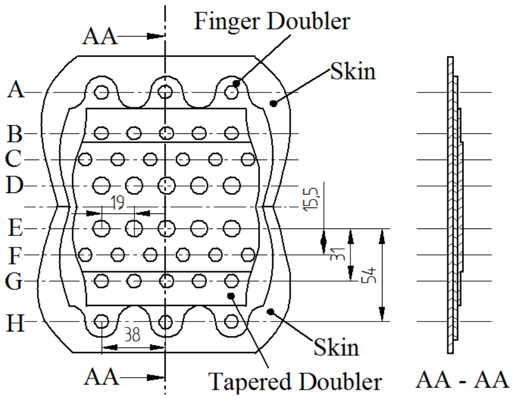


**Fig. 1: Circumferential joint configuration**

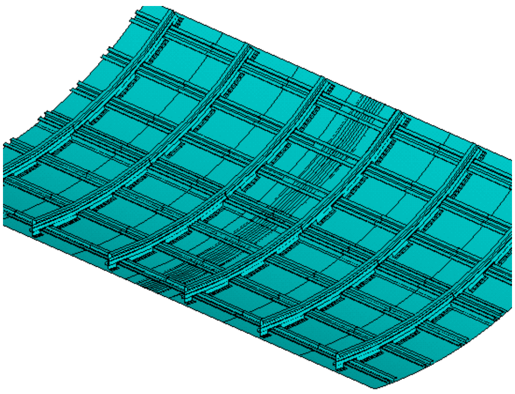

**Fig. 2: Curved panel considered for analysis**

**Table 1: Material property data**

| Part name | t (mm) | Material | E (MPa) | ν |
|---|---|---|---|---|
| Skin | 1.6 | Al 2024-T3 | 72395 | 0.33 |
| Stringer - Middle | 1.8 | Al 7075-T3 | 71016 | 0.33 |
| Stringer | 1.6 | Al 7075-T3 | 71016 | 0.33 |
| Shear Clip | 1.6 | Al 7075-T5 | 71016 | 0.33 |
| Frame | 1.8 | Al 7075-T4 | 71016 | 0.33 |
| Finger Doubler | 3.1 | Al 2024-T3 | 72395 | 0.33 |
| Tapered Doubler | 0.6-1.6 | Al 2024-T3 | 72395 | 0.33 |

**Table 2: List of joint regions in the model (see Fig. 1 for Row)**

| Joint Between | Location |
|---|---|
| Stringer and Skin | |
| Stinger and Shear clip | |
| Skin and Shear clip | |
| Shear clip and Frame | |
| Skin and Finger Doubler | Row – A |
| Skin and Finger Doubler | Row – H |
| Skin, Finger Doubler and Taper Doubler | Row – B |
| Skin, Finger Doubler and Taper Doubler | Row – C |
| Skin, Finger Doubler and Taper Doubler | Row – G |
| Skin, Finger Doubler and Taper Doubler | Row – D |
| Skin, Finger Doubler and Taper Doubler | Row – E |
| Skin, Finger Doubler and Taper Doubler | Row – F |

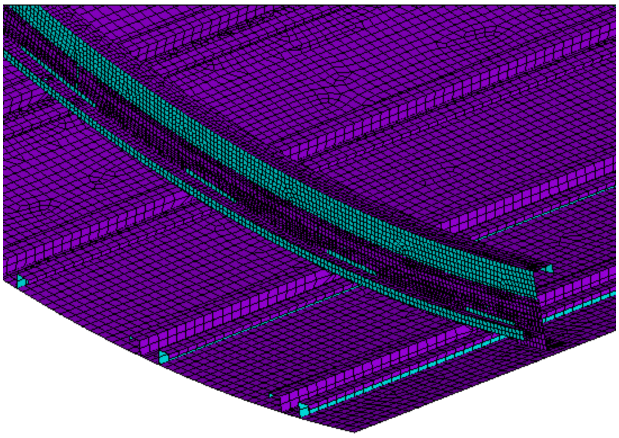

**Fig. 3: FE model of curved panel**

## 3. Results & discussions

Symmetric boundary condition is considered for all four edges of the meshed FE model. Pressure of 0.06937 MPa is applied over the skin surface to simulate internal pressure loading. Figs. 4 and 5 show the contour plots of resultant displacements. Maximum displacement occurs in the skin between stringers and ring frames. Figs. 6 and 7 show the contour plots of von Mises stress at the mid plane of the skin panel, frame and stringers. Maximum stress of 343.33 MPa is observed in a small area near the joint region between frame and stringer.

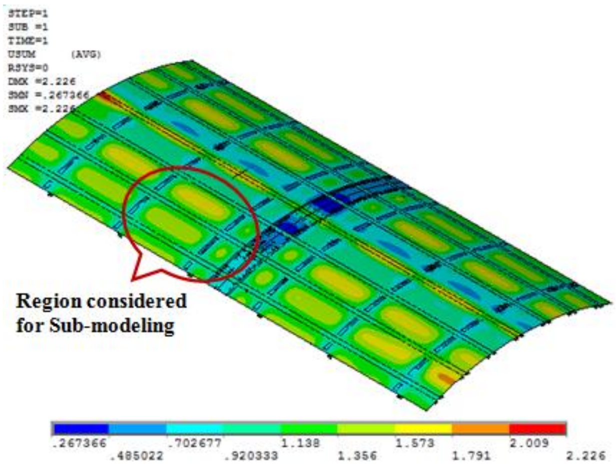


**Fig. 4: Resultant displacement contour**

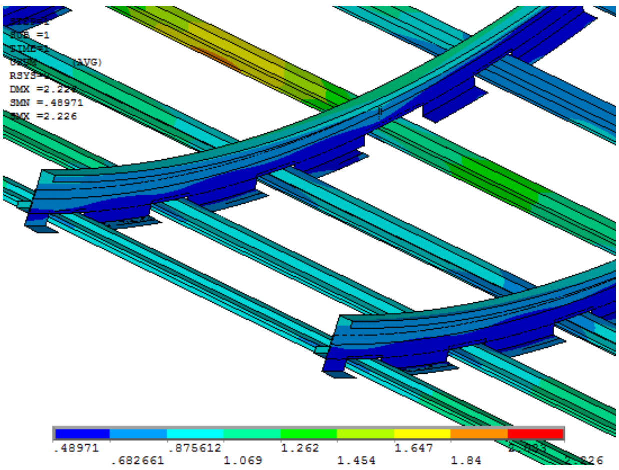


**Fig. 5: Max. displacement contour near circumferential joint**

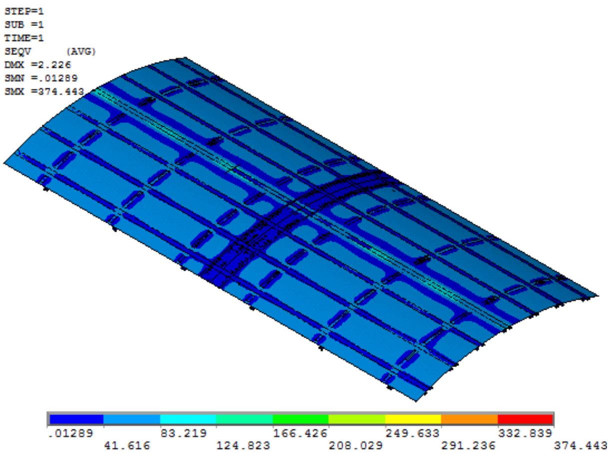


**Fig. 6: von Mises stress contour**

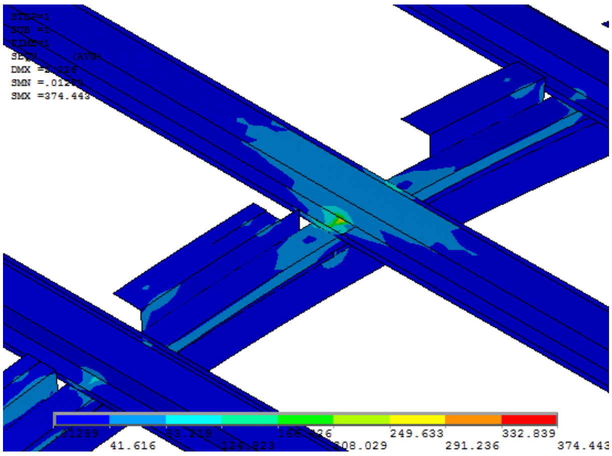

**Fig. 7: Max.von Misses stress contour at stringer & frame joint**

In the experimental investigation reported in [4] strain gauges were fixed on frames as shown in Fig. 8. The computed strains from this study are compared with the measured strain values and given in Table 3. The agreement is good enough to accept the proposed FE model for further investigations.

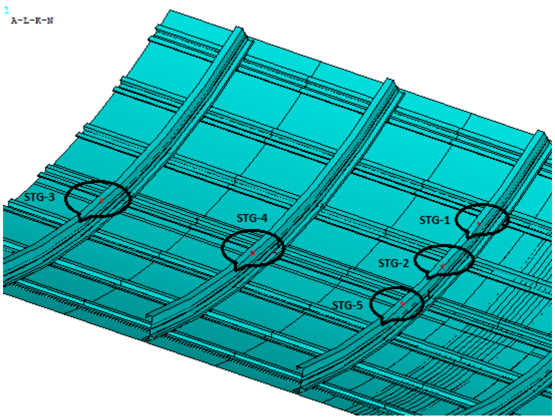


**Fig. 8: Strain gauge locations**

**Table 3: Comparison of measured and predicted strains**

| Strain Gauge No. | Experiment (με) | Analysis (με) | % Error |
|---|---|---|---|
| STG1 | 357.03 | 303.40 | 15.02 |
| STG2 | 404.02 | 292.02 | 27.72 |
| STG3 | 488.66 | 293.29 | 39.98 |
| STG4 | 508.60 | 310.65 | 38.92 |
| STG5 | 415.57 | 302.39 | 27.23 |

A hypothetical through thickness crack parallel to fuselage axis in the skin panel is assumed for fracture mechanics analysis. To obtain better solutions in the area of interest, sub-modeling capability available in ANSYS software is used. The selected region for sub-modeling is shown in Fig. 4. Fig. 9 shows an axially oriented through thickness crack in the skin panel. Fig. 9 shows the contour plot of resultant displacement obtained using sub-modelling. Good agreement in results can be seen comparing with results of complete model. Using an appropriate FE model of this cracked panel, stress distribution around crack tip can be determined. Contour plot of von Mises stress, plastic zone shape and size around each crack tip are presented in Fig. 10.

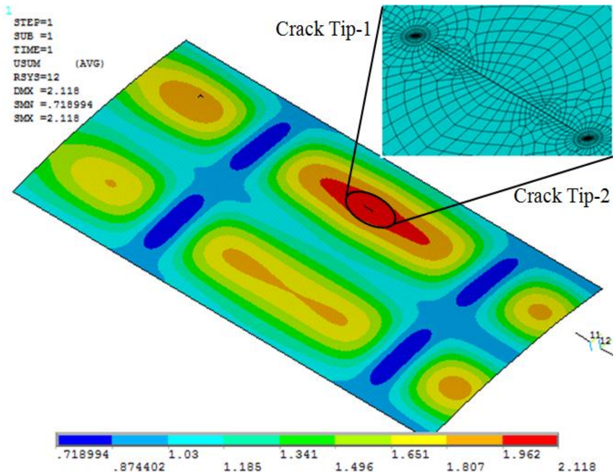


**Fig. 9: Displacement contour using sub-modeling and FE model of crack**

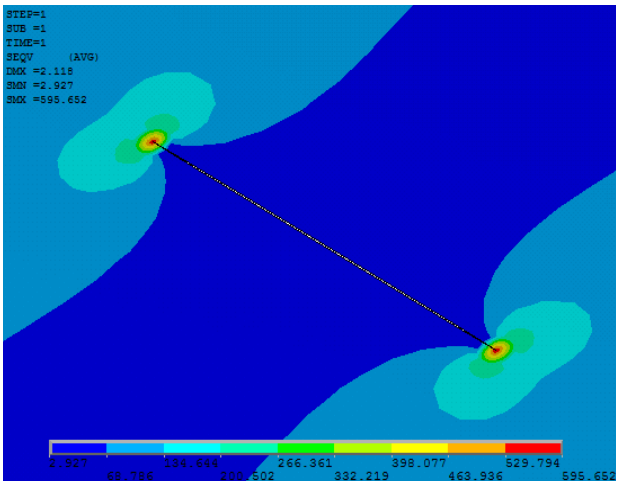


**Fig. 10: von Mises stress contour around an axial crack tips**

## 4. Conclusions

Fracture mechanics analysis of fuselage structural joints is intractable using analytical methods. Experimental investigations are prohibitively complex. The complex model was developed using SHELL 281 type available in commercial FEA software (ANSYS). The results presented in this paper validate the modeling and analysis capability in the ANSYS software for fracture mechanics analysis of aircraft fuselage structures including the effects of joints. The graphical post-processing capability in ANSYS software enables visualization of crack tip plastic zone shape and size and its variation along the crack front. Validation of FE simulation predictions using experimental data is a novel feature of this study.

## REFERENCES:


[1] S. Pitt and R. Jones. 1997. Multiple-site and widespread fatigue damage in ageing aircraft, *Engineering Failure Analysis*, 4(4), 237-257. http://dx.doi.org/10.1016/S1350-6307(97)00020-4.

[2] G. Samavedam, D. Hoadley and J. Davin. 1991. Test facility for evaluation of structural integrity of stiffened and jointed aircraft curved panels, *Structural Integrity of Aging Airplanes, Springer Series in Computational Mechanics*, 321-337.

[3] G. Samavedam and D. Hoadley. 1994. Fracture and fatigue strength evaluation of multiple site damaged aircraft fuselages - Curved panel testing and analysis, Final Report, DOT/FAA/CT-94/10.

[4] J. Bakuckas. 2002. Full-Scale Testing and Analysis of Fuselage Structure Containing Multiple Cracks, Report DOT/FAA/AR-01/46.

[5] R. Jones, L. Molent and S. Pitt. 1999. Study of multi-site damage of fuselage lap joints, *Theoretical and Applied Fracture Mechanics*, 32(2), 81-100. http://dx.doi.org/10.1016/S0167-8442(99)00029-4.

[6] D.Y. Jeong, O. Orringer and G.C. Sih. 1995. Strain energy density approach to stable crack extension under net section yielding of aircraft fuselage, *Theoretical and Applied Fracture Mechanics*, 22(2), 127-137. http://dx.doi.org/10.1016/0167-8442(94)00053-4.

[7] G.R. Irwin. 1957. Analysis of stresses and strains near the end of a crack traversing a plate, *Trans. ASME J. Applied Mechanics*, 24, 361-364.



**EDITORIAL NOTES:**

*Edited paper from International Conference on Advanced Design and Manufacture, 5-7 December 2014, Tiruchirappalli, Tamil Nadu, India.*

*GUEST EDITORS: Dr. T. Ramesh and Dr. N. Siva Shanmugam, Department of Mechanical Engineering, National Institute of Technology, Tiruchirappalli, Tamil Nadu, India.*