

Fracture Mechanics Analysis of Fuselage Structural Joints with Cracks

H. Ramesha, N. C. Mahendra Babu and H. V. Lakshminarayana

Abstract Fracture mechanics provides a methodology for prediction, prevention and control of fracture in materials, components, and structures. Fracture mechanics analysis is the backbone of damage tolerant design. Objectives of fracture mechanics analysis are: (1) stress analysis of cracks to derive crack tip stress field equations and define crack tip stress intensity factors, (2) determination of SIF solutions as function of crack length, orientation and applied loads for a given geometry, (3) prediction of mixed mode fracture under static, dynamic, and sustained loads, (4) prediction of residual strength as a function of crack length. The focus of this paper is on fracture mechanics analysis of longitudinal and circumferential joints of aircraft fuselage structure with cracks. The loading is by internal pressure. Commercial FEA software ANSYS and a special purpose post-processing subprogram called 3MBSIF are used to determine mixed mode membrane and bending SIF solutions. Residual strength prediction is based on the use of strain energy density theory of fracture. Significant results of this study are graphically presented and discussed in this paper.

Keywords Fuselage • Structural integrity • Longitudinal joint • Circumferential joint • Crack • Residual strength • Stress intensity factor

Nomenclature

t	Thickness
E	Modulus of elasticity
ν	Poisson's ratio
a_0	Initial half crack length

H. Ramesha (✉) • H. V. Lakshminarayana
Department of Mechanical Engineering, Dayananda Sagar College of Engineering,
Bangalore 560078, India
e-mail: bharatavarsh@gmail.com

N. C. Mahendra Babu
Department of Mechanical and Manufacturing Engineering, MSRUAS,
Bangalore 560058, India

a_{cr}	Critical half crack length
K_1	Mode 1 stress intensity factor
K_2	Mode 2 stress intensity factor
K_3	Mode 3 stress intensity factor
ΔK_1	Mode 1 stress intensity factor range
ΔK_2	Mode 2 stress intensity factor range
N	Number of cycles

1 Background

Fuselage of commercial transport aircraft is a complicated structure designed to withstand aircraft loads. Parts and assembly need to be designed to give more life to the structure there by reducing high replacement costs. Nowadays, more research is going on in extending the life of aircraft beyond its designed life. Fracture mechanics analysis helps in this aspect by determining the structural integrity, remaining life, and residual strength of the structure.

Longitudinal and circumferential joints formed on fuselage structure are more susceptible to fatigue damage [1]. As aircraft undergoes pressurization cycles, the parts are subjected to cyclic load which leads to initiation and growth of cracks at joint regions. After number of flight cycles, these cracks coalesce and will lead to failure of the structure [2]. Inspection of structure using NDT tests and damage tolerance analysis helps in analyzing the structure ensuring the safety of passengers and aircraft.

Experimental investigation of fuselage structure is a costly, tedious, and time-consuming process. However, in the absence of suitable reliable analytical means, many such investigations are carried out throughout the world, and a few of these results are also available in open literature [3, 4].

In this work, effort is made for fracture mechanics analysis of curved stiffened panels with longitudinal and circumferential joint using commercial FEA software ANSYS.

2 Problem Statement

Longitudinal and circumferential joints of fuselage structure are considered for fracture mechanics analysis. The specific problem was chosen for investigation, because complete geometry, dimensions, material property, applied loads, static test results, fracture test results, and fatigue crack growth test results are readily available in a report [4]. This is essential for validation and verification of predictions made in this study.

Analysis is carried out in following steps

- Stress analysis
- Evaluation of crack tip stress intensity factors
- Prediction of residual strength

2.1 Longitudinal Joint

A segment of fuselage structure shown in figure is considered for the analysis. Figure 1a shows details of the joint, consisting of outer skin, inner skin, outer doubler plate, inner doubler plate, and stringer bonded together. Evidently, the structure is quite complicated demanding finite element modeling for fracture mechanics analysis.

Figure 1b shows views of curved panel with lead crack. The assembly detail of parts is also shown in the same Fig. 1. A lead crack in the skin panel at joint row A is considered. Fracture mechanics analysis of the panel for various crack lengths is the main objective of this work.

2.2 Circumferential Joint

Figure 2a shows the details of a circumferential joint configuration in a fuselage structure consisting of skin, finger doubler plate, and tapered doubler plate. Location of starter crack can be observed between frame assemblies as shown in Fig. 2b..

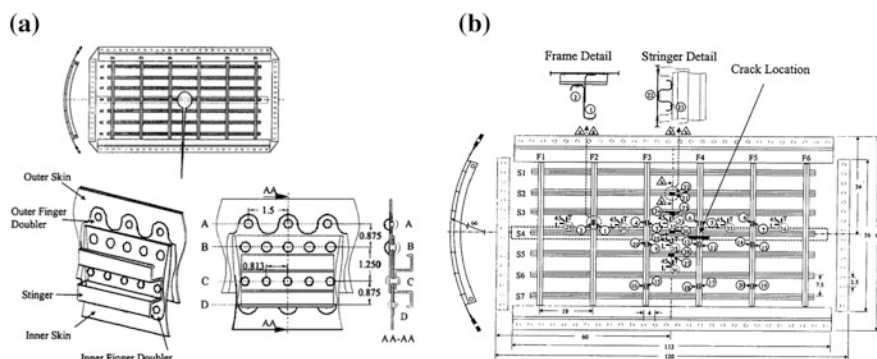


Fig. 1 **a** Longitudinal joint configuration; **b** curved panel with a starter crack [4]

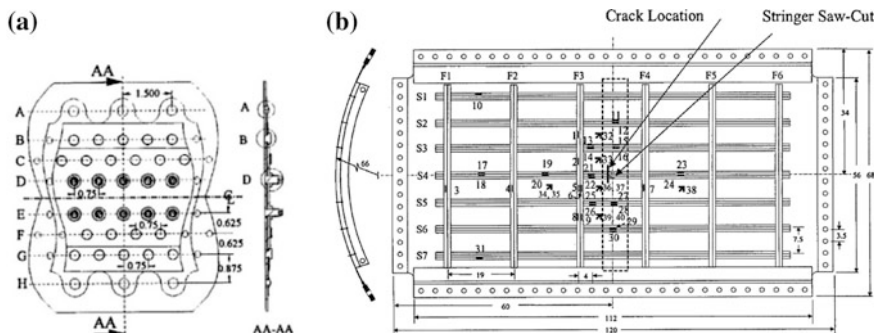


Fig. 2 **a** Circumferential joint configuration; **b** Circumferential joint in a curved panel with a starter crack [4]

A segment of Circumferential joint with seven stringers, six frame assemblies, two skin layers, and doubler plates shown in Fig. 2b, is considered for the analysis [4]. A lead crack in the skin panel at joint row A is considered. Fracture mechanics analysis of the panel for various crack lengths is the main objective of this work. Analysis of the curved stiffened panel is carried out in following steps

- FEA of the curved panels (without crack)
- Validation of results with available experimental data
- Fracture mechanics analysis of the curved panels

3 FE Modeling for Stress Analysis and Validation

3.1 Introduction

Finite element modeling is defined here as the analyst's choice of material models (constitutive relations and failure criteria), finite elements (types, shapes, orders), meshes, constraint equations, analysis procedures, governing matrix equations and their solution methods, specific pre- and post-processing options available in a chosen commercial FEA software (ANSYS) for the intended analysis (LEFT analysis) of the candidate component. The finite element model is developed as per dimensions and material property data extracted from reference [4]. Analysis is carried out for internal pressure of 0.06937 N/mm^2 with symmetric boundary conditions on all four sides. Thickness and material property data of curved panel components are shown in Table 1.

Table 1 Material property and thickness of parts

Part name	<i>t</i> (mm)	Material	<i>E</i> (MPa)	<i>ν</i>
Skin	1.6	Al 2024-T3	72,395	0.33
Stringer–middle	1.8	Al 7075-T3	71,016	0.33
Stringer	1.6	Al 7075-T3	71,016	0.33
Shear clip	1.6	Al 7075-T5	71,016	0.33
Frame	1.8	Al 7075-T4	71,016	0.33
Finger doubler	3.1	Al 2024-T3	72,395	0.33
Tapered doubler	0.6–1.6	Al 2024-T3	72,395	0.33

3.2 Longitudinal Joint

The geometry used to compute stress distribution and deformation is presented in Figs. 3a, b, and 4. Evidently, it is a very tough task even with the best preprocessor capability available in ANSYS.

For the specific panel under investigation, measured strains using strain gages for various applied pressure loads are reported in reference [4]. The measured and computed strain on frame assembly near the joint region is presented in Table 2. The agreement is good enough to accept the proposed FE model for further investigations.

Figure 5a displays the resulting deformations at the maximum pressure. Figure 5b presents contour plots of von Mises equivalent stress for the whole panel and identifies a maximum value of 485.77 MPa for internal pressure of 0.06937 N/mm². The graphical post-processing capability in ANSYS enables us to locate the region where it occurs as a reentrant corner in a frame close to the maximum deformation region in the panel.

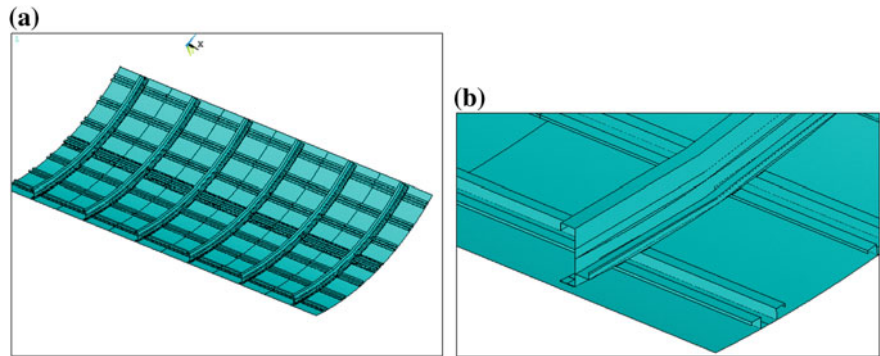


Fig. 3 **a** Curved panel considered for analysis; **b** assembly showing frame, stringer, and skin panel

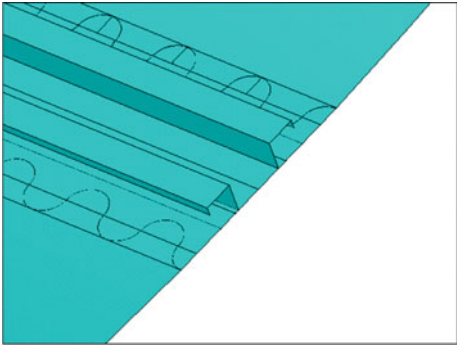


Fig. 4 Assembly showing stringer, doubler plates, outer and inner skin panels

Table 2 FE model validation using measured strain

Strain ($\mu\epsilon$) experiment [4]	Strain ($\mu\epsilon$) analysis	% Error
605.4	586.02	3.201

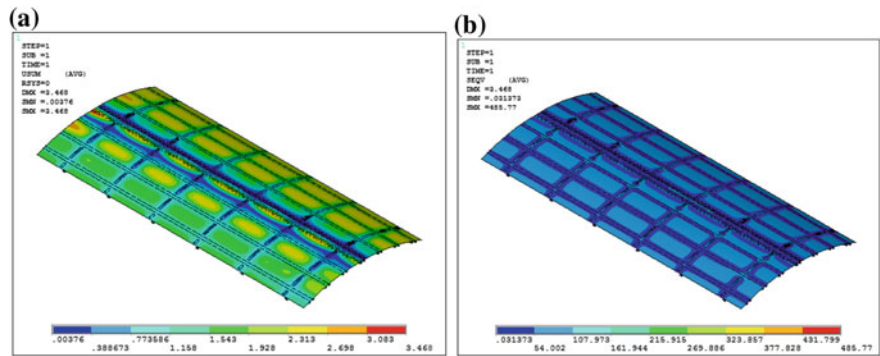


Fig. 5 **a** Contour plots of resultant displacement; **b** von Mises stress contour

3.3 *Circumferential Joint*

The geometry used to compute stress distribution and deformation is presented in Figs. 6a, b, and 7 highlights circumferential joint region of the panel. The computed and measured strain value on frame assembly near the joint region is presented in Table 3. The proposed FE model is used for further investigations.

Figure 8a displays the resulting deformations for internal pressure of 0.06937 N/mm². Figure 8b presents contour plots of von Mises equivalent stress for the whole panel and identifies a maximum value of 488.434 MPa in stringer near frame assemblies.

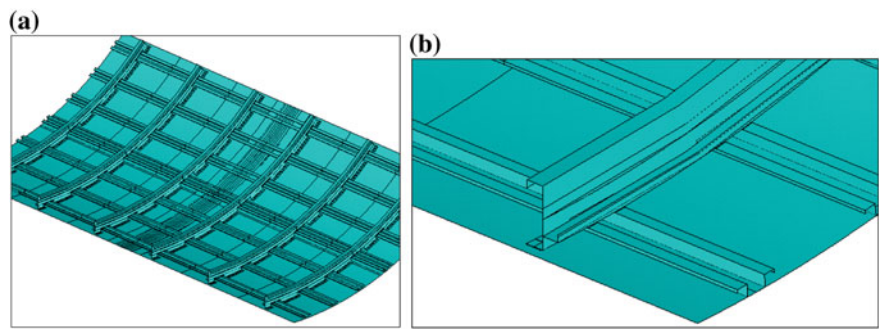


Fig. 6 **a** Curved panel considered for analysis; **b** assembly showing frame, stringer, and skin panel

Fig. 7 Circumferential joint

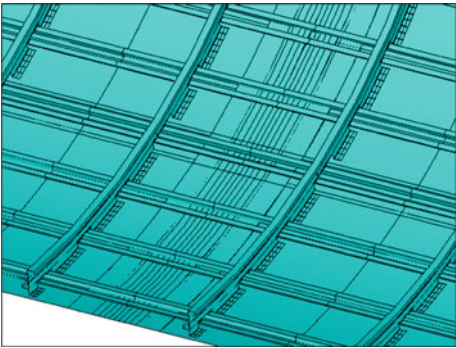


Table 3 FE model validation using measured strain

Strain ($\mu\epsilon$) experiment [4]	Strain ($\mu\epsilon$) analysis	% Error
357.03	303.4	15.02

4 Fracture Mechanics Analysis

4.1 Sub-Modeling and Computation of SIF's

4.1.1 Longitudinal Joint

ANSYS sub-modeling capability is used for evaluation of stress distribution and stress intensity factors at crack tips. Figure 9a shows the sub-model region considered for fracture mechanics analysis. Location of longitudinal lead crack in the curved panel can be observed from Fig. 9b.

It is to be noted that a single element type namely eight noded quadrilateral element quadratic in order (SHELL 281) is exclusively used. Around each crack tip, a refined mesh of singular elements (S-TRIA 6) with $NS = 20$, $\Delta a = a/50$, and ‘a’

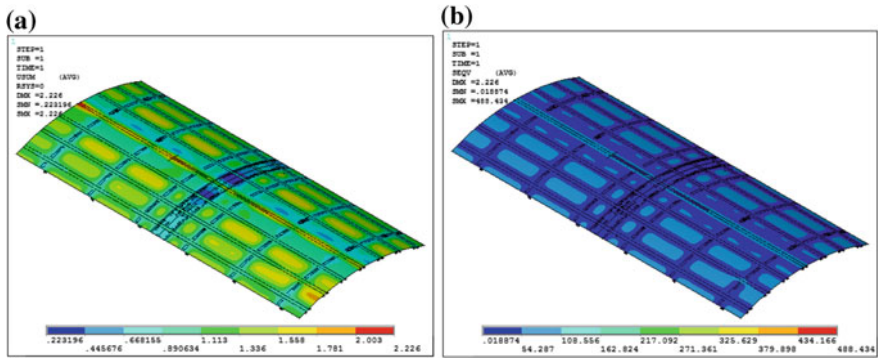


Fig. 8 **a** Contour plots of resultant displacement; **b** von Mises stress contour

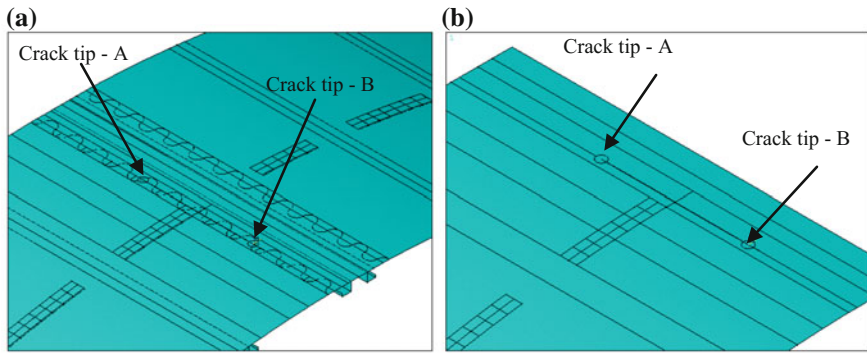


Fig. 9 **a** Sub-model for detailed analysis; **b** modeling of crack in outer skin panel (rivet row A)

is half crack length, and a compatible mesh of regular elements (QUAD 8 and TRIA 6) is used. The pre-processing command KSCON is exclusively used here. Parametric design language capability is used to model cracks of different lengths. Macros are written to generate FE models for given crack lengths.

Figure 10a, b displays von Mises stress contours around crack tips A and B. It is possible to identify the mode of fracture and estimate crack tip plastic zone shape and size from these figures. These data are essential for the choice of fracture criteria and prediction of fracture pressure and crack growth directions.

ANSYS software provides a post-processing command KCALC to compute mixed mode Stress Intensity Factors (K_I , K_{II} , K_{III}). The computed SIFs are presented in Fig. 11a, at the maximum pressure equal to 0.06937 N/mm^2 . These results are essential for the prediction of residual strength. Figure 11b provides ΔK values predicted for load range (Minimum/Maximum) equal to 0.1.

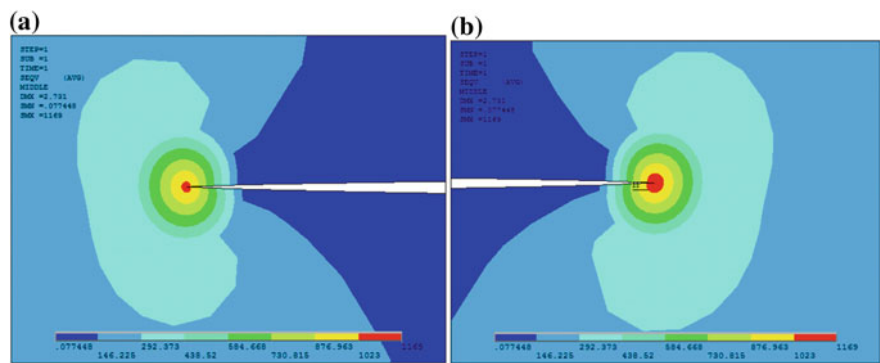


Fig. 10 **a** von Mises stress contour at crack tip—A; **b** von Mises stress contour at crack tip—B

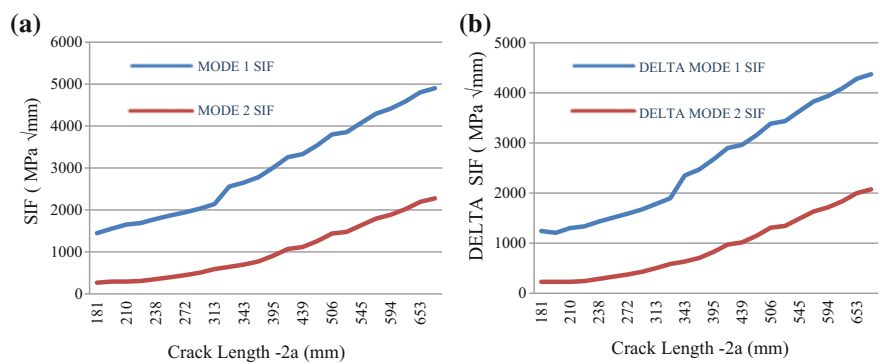


Fig. 11 **a** Stress intensity factor as a function of crack length; **b** stress intensity factor range as a function of crack length

4.1.2 Circumferential Joint

ANSYS sub-modeling capability is used for evaluation of stress distribution and stress intensity factors at crack tips. Figure 12a shows the sub-model region considered for fracture mechanics analysis. Location of circumferential lead crack in the curved panel can also be observed in the same figure.

Figure 13a, b displays von Mises stress contours around crack tips A and B. It is possible to identify the mode of fracture and estimate crack tip plastic zone shape and size from these figures. These data are essential for the choice of fracture criteria and prediction of fracture pressure and crack growth directions.

ANSYS software provides a post-processing command KALC to compute mixed mode Stress Intensity Factors (K_1 , K_2 , K_3). The computed SIFs are presented in Fig. 14a, at the maximum pressure equal to 0.06937 N/mm². These results are essential for the prediction of residual strength. Figure 14b provides ΔK values predicted for load range (Minimum/Maximum) equal to 0.1.

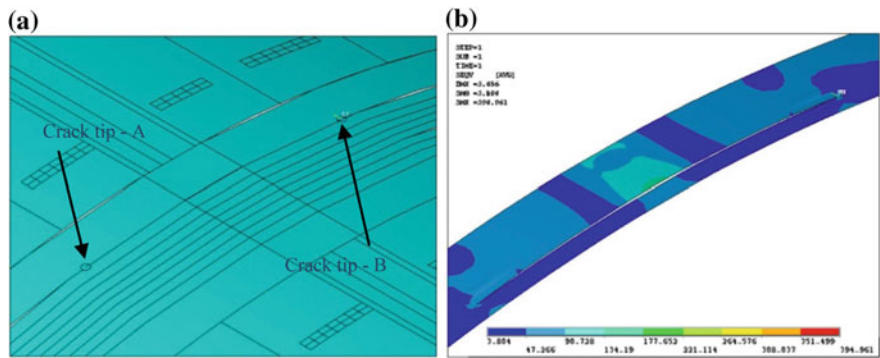


Fig. 12 **a** Sub-model for detailed analysis; **b** von Mises stress contour in the skin panel near crack location

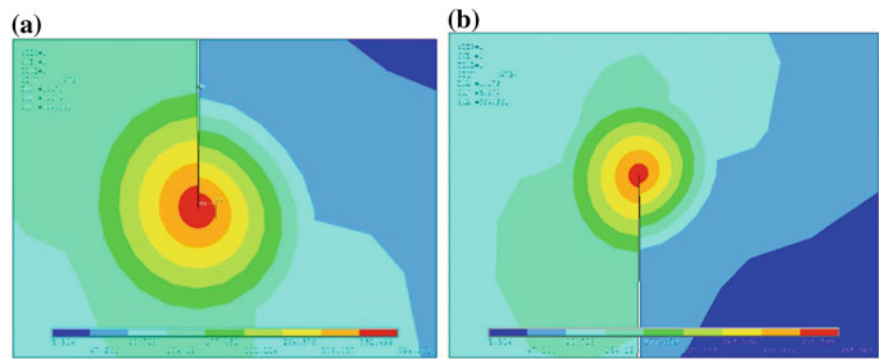


Fig. 13 **a** von Mises stress contour at crack tip—A; **b** von Mises stress contour at crack tip—B

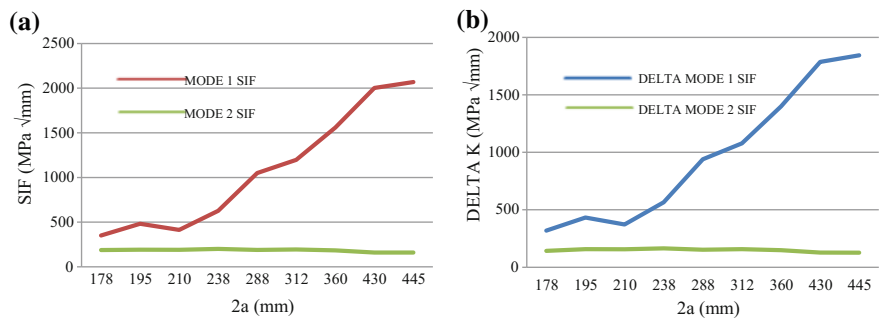


Fig. 14 **a** Stress intensity factor as a function of crack length; **b** stress intensity factor range as a function of crack length

4.2 Residual Strength Prediction and Correlation with Test Data

This section presents prediction of residual strength for longitudinal and circumferential joint curved panels using strain energy density theory of fracture. According to this crack extension will occur in the direction of minimum strain energy density ($S_{\min}$), and the extension will occur when it reaches a critical value (S_{cr}) which is a material dependent parameter [5].

4.2.1 Longitudinal Joint

Figure 15a shows variation of strain energy density factor with respect to crack length. Critical SED factor for skin panel material is 30.072. From the figure, it is evident that Minimum SED factor reaches critical value for crack length equal to 585 mm, which can be termed as critical crack length. From this analysis, it is found that catastrophic failure of the structure occurs when lead crack at the longitudinal joint region grows to its critical length of 585 mm. Figure 15b shows variation of residual strength for different crack lengths.

4.2.2 Circumferential Joint

Figure 16a shows variation of strain energy density factor with respect to crack length. Its value gradually increases to 17.12 for crack length equal to 550 mm on further increase in crack length SED factor deviates from its path and remained constant as shown. This is due to stringer underneath the crack which resists its growth. So crack will not extend unless this stringer fails. Similar observation is

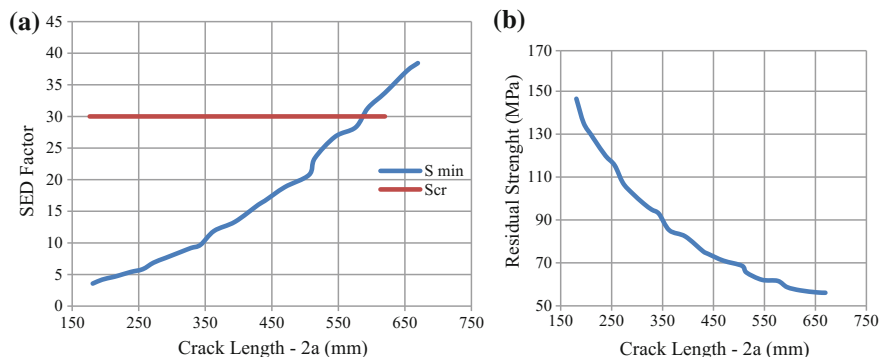


Fig. 15 **a** Variation of SED factor for different crack lengths; **b** variation of residual strength for different crack lengths

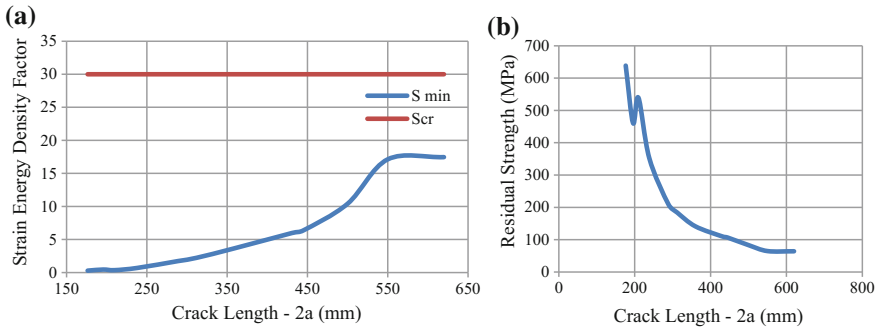


Fig. 16 **a** Variation of SED factor for different crack lengths; **b** variation of Residual strength for different crack lengths

found in experimental investigation, crack grows steadily till 560 mm and on further increase of load, the stringer underneath the crack breaks. Figure 16b shows variation of residual strength for different crack lengths. At crack length equal to 210 mm, a slight increase in residual strength can be observed, which is due to resistance by stringers located on either side of the crack.

References

1. S. Pitt, R. Jones, Multiple-site and widespread fatigue damage in ageing aircraft. *Eng. Fail. Anal.* 4(4), 237–257 (1997) (Elsevier)
2. G. Samavedam, D. Hoadley, Fracture and fatigue strength evaluation of multiple site damaged aircraft fuselages—curved panel testing and analysis. Final Report, DOT/FAA/CT-94/10, DOT-VNTSC-FAA-93-8 (1994)
3. G. Samavedam, D. Hoadley, J. Davin, in *Structural Integrity of Ageing Airplanes*, Test Facility for Evaluation of Structural Integrity of Stiffened and Jointed Aircraft Curved Panels, (Springer, 1991), pp. 321–337
4. J. Bakuckas, Full-scale testing and analysis of fuselage structure containing multiple cracks. Report DOT/FAA/AR-01/46, (2002)
5. D.Y. Jeong, O. Orringer, G. C. Sih, Strain energy density approach to stable crack extension under net section yielding of aircraft fuselage. *Theor. Appl. Fract. Mech.* 127–137 (1995) (Elsevier)

Advances in Structural Integrity

Proceedings of SICE 2016

Prakash, R.; Jayaram, V.; Saxena, A. (Eds.)

2018, XVII, 702 p. 468 illus., 244 illus. in color.,

Hardcover

ISBN: 978-981-10-7196-6