INTERNATIONAL JOURNAL OF ENGINEERING TECHNOLOGY AND SCIENCES (IJETS) Vol. 2 (1) DEC 2014
DOI: http://dx.doi.org/10.15282/ijets.2.2014.1.7.1014

Stress Analysis and Damage tolerance evaluation for wing structure with a large cutout in the bottom skin

Balaji Pukale
PG Scholar
Department of Mechanical Engineering
Malla Reddy College of Engineering & Technology,
Hyderabad, India
Email-balapukale@gmail.com

M.Amarnadha Reddy
Associate Professor & HOD
Department of Mechanical Engineering
Malla Reddy College of Engineering & Technology,
Hyderabad, India
Email-amarnadha@gmail.com

Abstract*— Wings are the lift generating components in the airframe structure. Wings are also used as fuel tanks in the transport aircraft. From the functional requirements point of view, cutouts are introduced in the bottom skin of the wing structure. Bottom skin is under tension stress field during flight. Cutouts in the bottom skin will act as stress risers due to stress concentration effect. The current project includes the stress analysis of a wing box with large cutouts in the bottom skin to identify the critical location for fatigue crack initiation. Damage tolerance evaluation includes the stress intensity factor calculations at crack tip. This is carried out by simulation of the crack in the finite element model. Stress intensity factor (SIF) at different crack lengths will be calculated using Modified Virtual Crack Closure Integral (MVCCI) method.

Keywords*— Damage tolerance design, Transport airframe, Stiffened panel, Finite element analysis,Wing box,SIF, MVCCI method.

I. INTRODUCTION

Aircraft is a flying machine which flies in air. Aircraft means it is not only airplane it will be categorized in to airplane, helicopters, parachute, and lighter-than-air vehicles. There are many aspects of design of aircraft structures are available. For modern jet aircraft, the design must incorporate clear aerodynamics shapes for long range flight near or at supersonic speeds, and or wings to open up like parachutes at very low speeds. Safety and weight are two major considerations in aircraft design. There should not be any compromise between safety and weight. Aircraft is mainly subjected to drag force, lift force, thrust force and weight due to these loads there are mainly five types of stresses occurred which are tensile stress, compressive stress, bending stress, shear stress and torsion stress. Frames and stiffeners will help to carry these loads. An aircraft is a complex structure, but a very efficient man-made flying machine.

The basic functions of an aircraft's structure are to transmit and resist the applied loads to provide an aerodynamic shape and to protect passengers, payload systems, etc., from the environmental conditions encountered in flight. Airframe design is a field of engineering that combines aerodynamics, materials technology, and manufacturing methods to achieve balances of performance, reliability and cost. Aircrafts are generally built-up from the basic components of wings, fuselage, tail units and control surfaces. Each component has one or more specific functions and must be designed to ensure that it can carry out these functions safely. Any small failure of any of these components may lead to a catastrophic failure of material.

a) Material Selection

Selection of aircraft materials depend on the considerations which are categorized as structural performance and cost. The key material properties that are required to maintain cost and structural performance are,

- Ultimate and Yield strengths
- Damage tolerance (fracture toughness and crack growth)
- Density
- Young's modulus
- Fracture toughness
- Thermal expansion
- Fatigue strength
- Corrosion

Based on the above considerations some other criteria associated with producing the basic materials in the form of requirement and fabricating the end product at a reasonable cost. They are

- Material cost
- Availability
- Fabrication characteristics

Airframe structural components are made of wide variety of materials based on their properties and requirement. The earliest aircrafts were constructed from wood. Now modified aircrafts are constructed from molded composite materials, such as carbon fiber. Structural members of an aircraft's fuselage include bulkheads, stringers, longerons, ribs, spars and more. The skin of aircraft is made from a variety of materials, ranging from impregnated fabric to plywood, aluminum, or composites. Under the skin and attached to the structural fuselage are the many components that support airframe function. The entire airframe and its components are joined by rivets, bolts, screws, and other fasteners. Welding, adhesives and special bonding techniques are also used.

Objective and Scope of the work

The main objective of this study is to evaluate the Stress analysis and Damage tolerance evaluation for wing structure with a large cutout in the bottom skin.

- Stress analysis of the wing structure with a large cutout in the bottom skin using FEA methodology.
- Refinement of the analysis model near the high stress region to capture the stress concentration and gradient stress field.
- Calculation of stress intensity factor for different crack length in the stiffened panel.

Methodology

Stress analysis: Stress analysis is carried by Finite element method.

Software used: MSC Patron & MSC Nastran is a finite element analysis (FEA) program that was originally developed for NASA in the late 1960s under United States government funding for the Aerospace industry. The MacNeal-Schwendler Corporation (MSC) was one of the principal and original developers of the public domain NASTRAN code. NASTRAN source code is integrated in a number of different software packages, which are distributed by a range of companies.

- **Stress intensity factor calculation:** SIF calculations carried by Modified Virtual Crack Closure Integral method (MVCCI).

II. PROBLEM FORMULATION

CAD modeling of the wing structure with a large cutout in the bottom skin is carried out using CATIA V5 tool. Stress analysis of stiffened panel with structure with a large cutout in the bottom skin is done by MSC PATRAN and NASTRAN for the internal pressurization load.

Scope of problem Statements

Steps used in MSC-Nastran and MSC-Patran is shown below

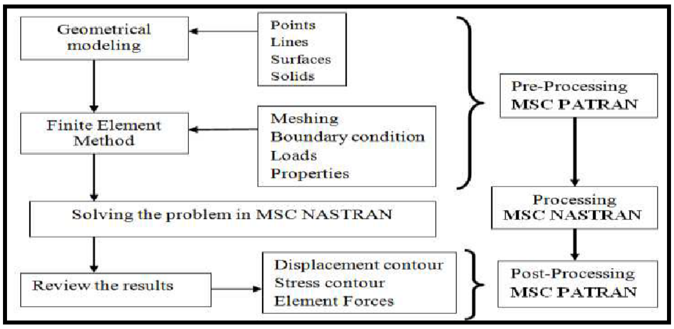

Fig: 1 Steps involved in problem formulation

Geometric modeling

A geometric model of the with a large cutout in the bottom skin is carried by CATIA V5 R20 with specified geometrical configuration available by using points, lines, surfaces and solids.

Finite Element Method

Elements, loads after the preparation of geometric model, it is imported in FEA software MSC-PATRAN and then extraction and meshing of model is done. After appropriate meshing using QUAD and TRIA and boundary conditions are applied at required locations of the model and then properties are defined to the various components and parts of fuselage structure and then analysis is done.

Solving

After running the analysis, the database file is imported in the solver MSC-NASTRAN software for the solving of the problem

Review

After solving the problem, the result file created by solver software is imported in MSC-PATRAN for reviewing the results. The results are carried out in the form of stress tensor over the whole structure and deformation of parts and full model.

Material Specification

Selection of aircraft materials depends on their properties which can be categorized on cost and structural performance. Mechanical properties of the skin, stiffening members and rivets are required for finite element models. There is little information on the material properties of skin, stiffening members and rivet material in the literature. Aluminum 2024 is used for components fuselage and rivet. Table 1 describes few material properties used for analysis.

Table 1 Material properties used for analysis

Material Property	Aluminium-2024 T3
Ultimate tensile strength	483 MPa
Tensile yield strength	362 MPa
Young's modulus	70000 MPa
Poisson's ratio	0.33
Fracture toughness	98

Aluminum alloys are widely used in modern aircraft construction because of their material properties shown in table above. There are also different grade among them aluminum 2024 and 7075 alloys are most commonly used. The 2024 alloys T3 have good fracture toughness. T is the code number which indicates the heat treatment process. The 7075 alloys (7075-T6, T6510) have higher strength than the 2024 but lower fracture toughness. Fracture toughness is very important parameter.

III. STRESS ANALYSIS

The CAD model prepared in CATIA V5 is imported in MSC PATRAN FEM software and then each curve surfaces are extracted for further working processes. The wing structure with a large cutout in the bottom skin is divided into groups according to their properties. The meshing includes quad and tria (shell) elements with fine mesh is done in between cutout edges. Corse meshing is done wherever fine meshing is not required. The cutout is associated with rivet holes which are present in actual cutout of bottom skin structure. For the meshing in skin, shell elements especially QUAD elements are used with fine mesh size around the cutout because the chances of getting stress concentration near the cutout is very high and also coarse meshing is done with both QUAD and TRIA elements. Coarse meshing is done for stringers where QUAD elements are used. While meshing them, it should be in mind that the nodes of these elements must also match with the nodes of skin where connectivity is achieved by 1-D bar element rivets. MPC (multi point constraint) is done for the rivets around the cutout which are used to apply more sophisticated constraints on the FEM model such as sliding boundary conditions.

Finite Elemnt Method of wing structure with large cutout at bottom skin

Meshing can be done using MSC-nastran and MSC-patran software. Fine meshing with 1mm elemental length is done around the cutout region. Course meshing is done away from the stress concentration region . Connectivity established between them by modifying the mesh.

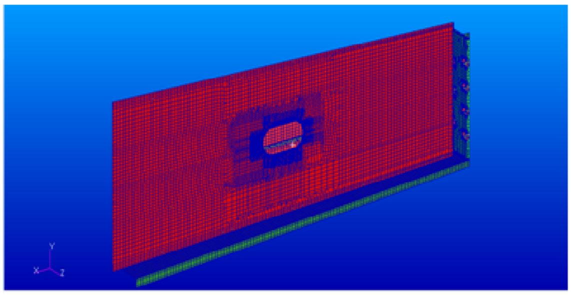

Fig: 2 Finite element of wing structure

Table 2 Total no. of nodes and elements are used

Product Description	Types of Elements	No. of Elements	No. of Nodes
Bottom Skin	QUAD4	11268	11708
	TRAI3	121	
Top Skin	QUAD4	5414	5579
	TRAI3	7	
Ribs & Spar	QUAD4	9680	11135
	TRAI3	223	
Stringers	QUAD4	3880	4704
Rivet	1D bar	1530	2836

Loads and Boundary conditions

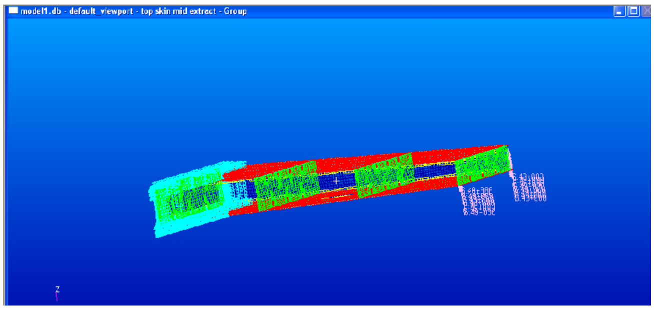

Fig: 3 shows load and boundary condition

Wing span =10.97m

Fuselage width=1.12m

Lift load /wing = 0.4*1362kg = 652.8kg

Total lift load/wing = 652.8*4.5 = 2937.6kg

L1 = 0.0895*2937.6 = 262.91

L2 = 0.1093*2937.6 = 321.07

L3 = 0.1243*2937.6 = 365.143

L4 = 0.1341*2937.6 = 393.93

L5 = 0.1391*2937.6 = 408.62

L6 = 0.134*2937.6 = 396.63

L7 = 0.12129*2937.6 = 356.301

L8 = 0.0895*2937.6 = 262.9152

L9 = 0.0398*2937.6 = 116.918

L10 = 0.0199*2937.6 = 58.458

Total Bending moment = 7009937.472kg-mm

MbTotal = 7*106 kg-mm

Bending moment = Force * Distance

Mb = F*D

F= 7*106/ 2800

F = 2503.54 kg

F/L = 2503.54/386 = 6.48 kg/mm

Deformation contour

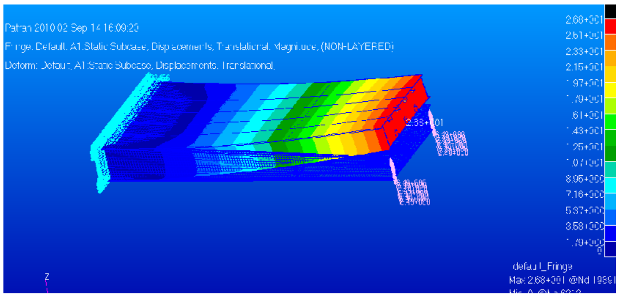

Fig: 4 The deformation in wing structure

Maximum deflection at the tip was found to be **26.8** mm

Stress Contour

The maximum stress is found at the rivet hole location in the skin near the cutout region. Where the rivets are used to fasten the stiffeners on the skin. Fine mesh was carried out to capture gradient stress distribution

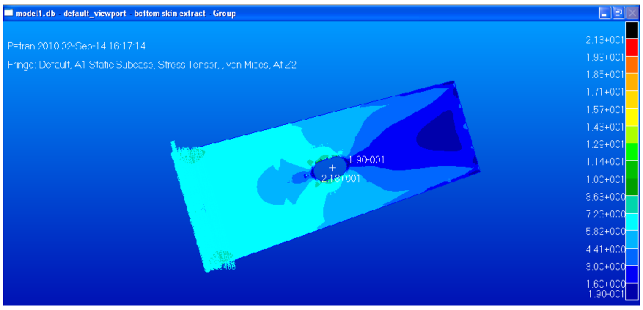

Fig: 5 Fig shows induced stress

Maximum stress was found near the rivet hole of fuel access cutout of magnitude **21.3 kg/mm^2**

IV. CALCULATIONS

SIF can be calculated by Modified Virtual Crack Closure Integral (MVCCI) method. SIF for 10 mm crack length is calculated by

$$K = \sqrt{G \times E}$$

Where,

K is the stress intensity factor in MPa√m

G is the strain energy release rate in N/mm

E is young's modulus in N/mm^2.

Strain energy release rate is calculated by

$$G = \frac{1}{2\Delta a} \times \Delta V \times \frac{F}{t}$$

Where,

G is strain energy release rate

F is grid point force balance

ΔV is the crack opening displacement

Δa is the elemental edge length near the crack tip in mm,

T is the thickness in mm,

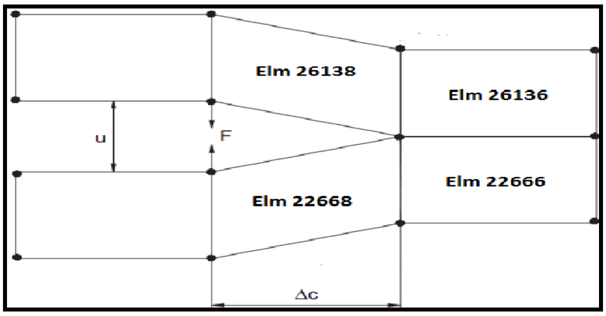

Fig: 6 Finite Element Models in The Vicinity Of a Crack Tip

The values of crack opening displacement and force at the crack tip are taken from F06 file after solving from software that is shown below. Displacement vector gives the values of crack displacement vector for node 679337is 0.3984 mm which is taken as V_2 that is shown below. Where T_1, T_2, T_3 are translations along X, Y, Z axis and R_1, R_2, R_3 are the rotations about X,Y,Z axis respectively.

SUBCASE 1

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
67930	G	-4.475648E-01	-2.682252E-01	0.0	2.526728E-06	0.0	-5.000824E-03
67931	G	-4.424221E-01	-2.675453E-01	0.0	3.267063E-06	0.0	-5.290046E-03
67932	G	-4.369550E-01	-2.667936E-01	0.0	4.262981E-06	0.0	-5.649447E-03
67933	G	-4.310673E-01	-2.659526E-01	0.0	5.436356E-06	0.0	-6.118123E-03
67934	G	-4.246006E-01	-2.650045E-01	0.0	6.435563E-06	0.0	-6.742304E-03
67935	G	-4.173965E-01	-2.638931E-01	0.0	4.350674E-06	0.0	-7.729006E-03
67936	G	-4.086406E-01	-2.625903E-01	0.0	-6.905727E-06	0.0	-9.126379E-03
67937	G	-3.984200E-01	-2.607655E-01	0.0	-8.575976E-05	0.0	-1.402569E-02
67938	G	-2.355811E-01	-2.334752E-01	0.0	3.504548E-09	0.0	2.565620E-03
67939	G	-2.381026E-01	-2.339406E-01	0.0	1.305968E-09	0.0	2.613875E-03

Fig: 7 (a) Crack displacement vector for node 67933

Every node has 4 elements; force in two upper elements will be equal to lower elements but in opposite direction. So, two elements are considered for calculating SIF. Element 49551 has force of 73.32 which is F2. And element 47058 has force of 83.96 which is shown below.

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
87494	G	-3.320012E-01	-2.519818E-01	-8.517950E-07	-8.698226E-06	4.558710E-06	7.971887E-03
87495	G	-3.408306E-01	-2.531206E-01	-2.062727E-05	1.821033E-06	-6.896138E-06	9.497511E-03
87496	G	-3.511046E-01	-2.545896E-01	-7.929532E-05	9.235145E-05	-2.590472E-05	1.511271E-02

THIS IS A DEFAULT SUBCASE. OCTOBER 18, 2014 MSC.NASTRAN 4/ 3/07 PAGE 467

SUBCASE 1

Fig: 7 (b) Crack displacement vector for node 87492.

SUBCASE 1

G R I D   P O I N T   F O R C E   B A L A N C E

POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
63079	60264	QUAD4	-8.998107E-02	4.355455E-01	2.408462E-03	-2.563004E-04	-4.181360E-04	4.378346E-06
63079	60265	QUAD4	1.027490E-01	-4.014699E-01	-2.536662E-03	2.347601E-04	-2.041061E-04	-4.035950E-06
63079	60266	QUAD4	1.364764E-01	-2.343311E-01	-2.001072E-03	1.787000E-04	-3.517481E-04	-4.532007E-06
63079		*TOTALS*	-5.730694E-13	-1.366351E-12	-7.979728E-16	-2.678413E-15	5.251875E-16	-1.541735E-16
63893		F-OF-SPC	0.0	0.0	9.028187E-03	0.0	7.272272E-01	0.0
63893	45516	QUAD4	7.121706E-01	-3.340939E+01	-4.657988E-01	-6.627645E-01	-2.858907E-01	2.062971E-02
63893	47058	QUAD4	-8.396934E+01	-7.818355E+01	6.849504E-01	7.017029E-01	-2.797285E-01	-2.137024E-02
63893	49404	QUAD4	8.607967E+01	6.303432E+01	-1.186125E+00	-4.593822E-02	-7.045837E-02	3.286006E-03
63893	49551	QUAD4	-7.332739E+01	4.855862E+01	9.579455E-01	6.999877E-03	-9.114964E-02	-2.545471E-03
63893		*TOTALS*	3.737455E-12	2.359002E-12	-6.661338E-15	-4.801715E-15	-1.817990E-15	-9.540979E-18
64005		F-OF-SPC	0.0	0.0	-1.608808E-01	0.0	4.024233E-01	0.0
64005	45516	QUAD4	-3.645241E-01	4.633610E+00	6.272202E-01	-2.619352E-01	-1.741858E-01	6.311510E-03
64005	45544	QUAD4	4.774560E-01	-2.267763E+01	-5.114263E-01	-1.182544E-01	-1.018729E-01	2.130523E-03

Fig: 7 (c) Force at tip of crack

Strain energy release rate is,

$$G = \frac{1}{2\Delta a} \times \Delta V \times \frac{F}{t}$$

Elemental length,

$\Delta a = 1$ mm

Crack opening displacement,

$\Delta V = V_1 - V_2$

$= 0.3984 - 0.3511$

$= 0.0473$ mm

Force at tip of crack,

$F = F_1 + F_2$

$= 73.32 + 83.96$

$= 157.28$ Kg

Thickness, t = 2mm

Young's modulus, E = 7000

Substituting these values in above equation,

$$G = \frac{1}{2 \times 1} \times 0.0473 \times \frac{157.28}{2}$$

Stress Intensity Factor is calculated by

Where,

G = 1.69 Kg/mm

$E = 7000\ kg/mm^2$

So,

$K = 108.76\ kg/mm^2 \sqrt{mm}$

$K = 108.76 \quad 9.81 \quad 10^{-3/2}$

$K = 33.74\ mpa\sqrt{m}$.

V. RESULTS

SIF can be calculated for different incremental crack lengths ranging from 10 mm to 637 mm. SIF for different crack length is shown in table below.

Table: 3 Stress intensity factor values for different crack lengths of the stiffened panel

ΔC	V_1	V_2	ΔV in mm	F_1	F_2	F in mm	Δa	G N/mm	K MPa$\sqrt{m}$
25	0.296	0.259	0.0367	87	68	155.03	1	0.89	24.49
50	0.276	0.318	0.0421	72	70	142.07	1	1.35	30.26
75	0.35	0.301	0.0486	83	82	161.62	1	1.78	34.7
100	0.372	0.32	0.0519	90	85	175.07	1	2.06	37.29
125	0.039	0.34	0.0517	90	84	174.15	1	2.04	37.12
127	0.395	0.344	0.0507	89	82	170.35	1	1.96	36.36
130	0.396	0.347	0.0495	87	79	165.89	1	1.86	35.45
132	0.398	0.351	0.0473	84	73	157.28	1	1.69	33.74

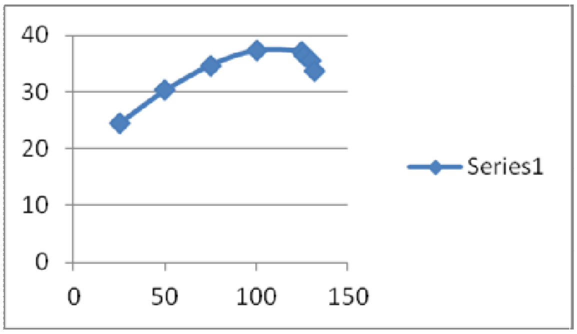

Fig: 8 SIF vs. crack length

VI. CONCLUSION

Damage tolerance design philosophy is generally used in the aircraft structural design to reduce the weight of the structure. The air load distribution was considered for the stress analysis of the wing structure. The crack is initiated from the location of maximum tensile stress. MVCCI method is used for calculation of stress intensity factor.FEM methodology is used for the stress analysis. A stiffened panel with a fuel access cutout was analyses with and without the presence of the crack. Variation of SIF is studied for different crack length. Plot indicates that SIF decreases near stiffener location. Residual strength calculation where carried out for different crack length. Residual strength plot indicates that the crack get arrested near the stiffener location. Results obtained from the current study are in good comparison with results shown from an International journal paper.

VII. AKNOWLEDGMENT

First and foremost, I offer my sincere gratitude to my internal guide Mr. Mr. AMARANADH REDDY.M, Associatie Professor & HOD of Mechanical Engineering who has supported me throughout this project with her/his patience and valuable suggestions. I would like to thank Mr. Y.Dilip Kumar, PG (M.Tech) Coordinator for his valuable guidance and encouragement during my dissertation work I am also grateful to the Principal Dr. V.S.K. Reddy for providing me with all the resources in the college to make my project a success.

VIII. REFERENCES

[1] Newman, J. C., Jr., "Finite Element Analyses of Fatigue Crack Propagation -- Including the Effects of Crack Closure", Ph.D. Thesis, Virginia Polytechnic institute and State University, Blacksburg, VA, May 1974.

[2] Newman, J.C., Jr., "An Elastic-Plastic Finite Element Analysis of Crack Initiation, Stable Crack Growth, and Instability", ASTM STP 833, 1984,

[3] W.T. Chow and S.N. Alturi, "Finite element calculations of stress intensity factors for interfacial crack using virtual crack closure integral", Computational mechanics 16 (1995).

[4] G.I. Nesterenko, B.G. Nesterenko, "Ensuring structural damage tolerance of russian aircraft", *International Journal of Fatigue,* pp. 1054–1061, 31 Jan 2009.

[5] Lucjan Witek , "stress intensity factor calculations for the compressor blade with half-elliptical surface crack using raju-newman solution" *Fatigue of Aircraft Structures*, Vol. 1, pp. 154-165, 2011.