Static stress analysis of a wing bottom using Finite Element Method

A.Animhons[1], N. Maheswaran[2], M.S.Prabhu[3], A.Sankaran[4]

[1,2,3] Assistant Professor, Department of Aeronautical Engineering & Hindusthan Institute of Technology, Coimbatore
[4] Professor& Head, Department of Aeronautical Engineering & Hindusthan Institute of Technology, Coimbatore
[1]animhonsasokan@yahoo.com

Abstract: Finite element method is widely used in the stress analysis of engineering structures. Aircraft is symbol of a high performance mechanical structure with a very high structural safety record. Rarely an aircraft will fail due to a static overload during its service life. As the aircraft continues its operation, fatigue cracks initiate and propagate due to fluctuating service loads. This project deals with the problem of identification of the critical location for fatigue crack initiation. Fatigue is a phenomenon by which the load carrying ability of a structure decreases when subjected to fluctuating loads. In a metallic structure fatigue manifests itself in the form of a crack which propagates. If the crack in a critical location goes unnoticed it could lead to a catastrophic failure of the airframe. In a structure like airframe, a fatigue crack will appear at the location of high tensile stress. Further these locations are invariably the sites of high stress concentration. Therefore, the first step in the fatigue design of an airframe is the identification of high tensile stress locations. This requires a global FEA of the structure. At the site of high tensile stress one has to identify one of the many stress concentrations present in that area. This is facilitated by a local refined FEA. This is followed by an estimation of the local stress at the highest stress concentrator. A wing panel with a large cutout in the bottom skin is considered for the analysis. Load corresponding to the level flight condition with maximum bending moment will be calculated. Local analysis near the cutout area will be carried out. Stress concentration factor and gradient stress field will be captured.

Keywords

Aircraft, Stress, wing bottom skin, large cutout, Fatigue crack, Stress concentration, Finite element method.

1. INTRODUCTION

Aircraft is symbol of a high performance mechanical structure, which has the ability to fly with a very high structural safety record. Aircraft experiences variable loading in service. Rarely an aircraft will fail due to static overload during its service life. The main lift generating components in the aircraft structure is the wings. About 80% of the lift load is taken by the wings. Another function of the wing is that they are also used as fuel tanks in commercial aircrafts. During flight, the lower part of the wing experiences a tensile stress and the upper part experiences a compressive stress. The current case considers the bottom or lower part of the wing, which experiences tensile stresses. In order to withstand the bending of the wing section due to transverse loads acting on the wing, the wing box is provided with integrated stiffeners. Cut out which is intended to provide passage for fuel access comprises of auxiliary holes around the small cut out. Discontinuities or flaws in any structure leads to high stress concentration at that region. Here, cut-out with auxiliary holes will be the critical region. These are probable locations for fatigue crack initiation. So stress analysis of that wing box is necessary.

2. OBJECTIVE

An aircraft wing is one of the major components. And the aircraft wing will be subjected to various loading cases. In that the most critical one will be lift loading condition. So if we solve the problem for lift load condition we can ensure the safety for all other load cases. So when lift load is acting on the wing, it will bend. When wing is subjected to bending, the upper surface will undergo compression and bottom surface will undergo tension.

3. FEA – FINITE ELEMENT ANALYSIS

The finite element method (FEM) is a numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimization. A domain of interest is represented as an assembly of finite elements. Approximating functions in finite elements are determined in terms of nodal values of a physical field which is sought. A continuous physical problem is transformed into a discretized finite element problem with unknown nodal values. For a linear problem, a system of linear algebraic equations should be solved. Values inside finite elements can be recovered using nodal values.

4. CONCEPTUAL DESIGN

The conceptual design of the fuel access cutout panel is modeled using commercially available modeling software CATIA V5. Model showing the fuel access cutout, which has skin, cutouts, holes, stiffeners. By using the NASTRAN and PATRAN software, meshing is done, and loads and boundary conditions are applied. Analysis is done, and hence we get the maximum stress located point.

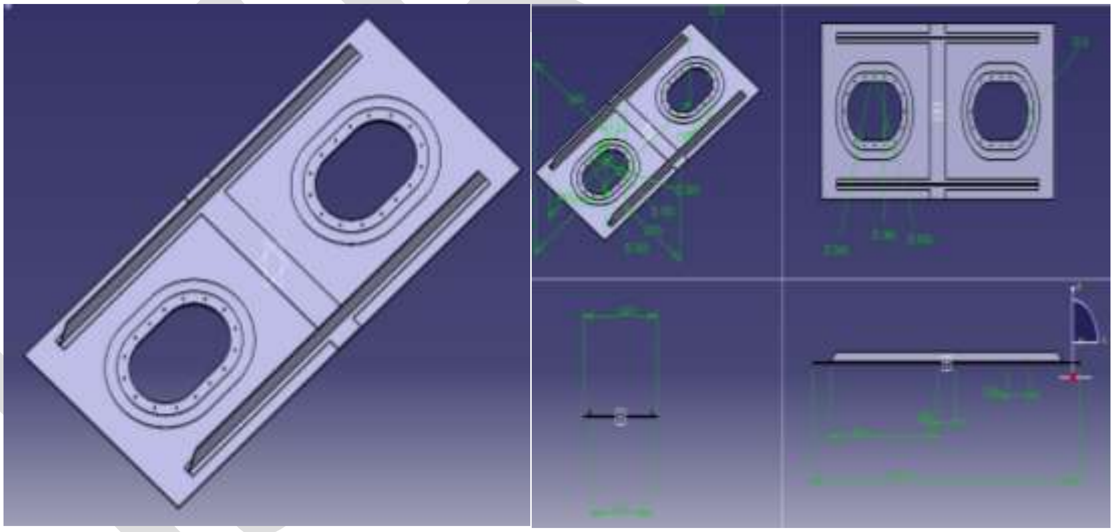


Fig.4.1 Conceptual design

5. COVERGENCE GRAPH

In order to determine the best element size which may give an accurate result will go for the convergence study. Here I selected the plate with hole in approximate dimensions, modeled, meshed, applied load and boundary conditions and analyzed.

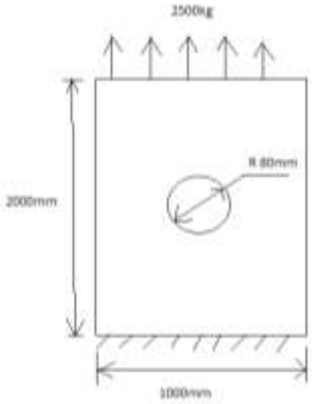


Fig.5.1 Plate with hole geometry

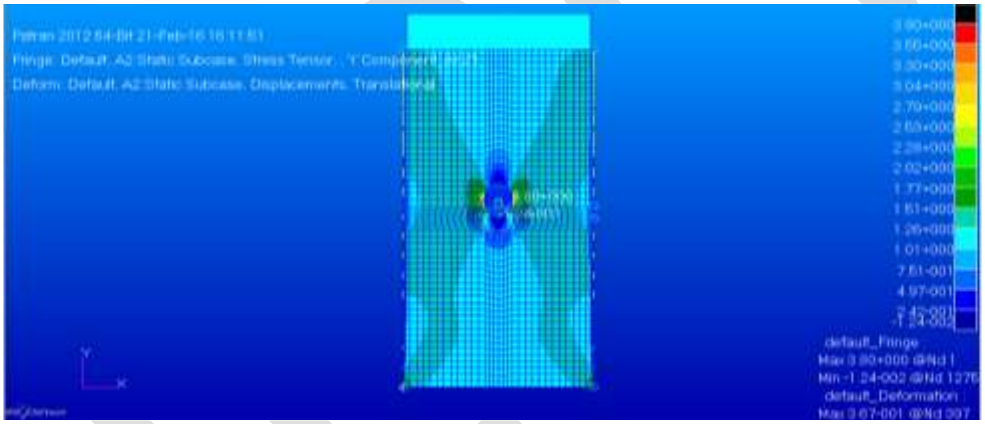


Fig.5.2 Analysis

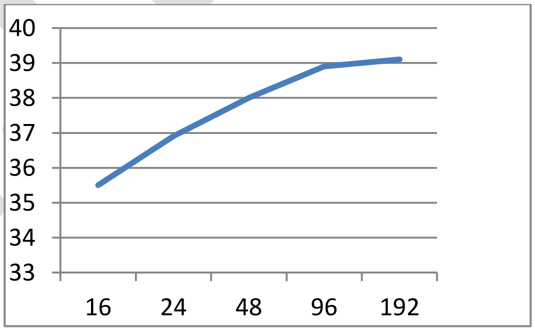


Fig.5.3 Convergence graph

X-axis - no. of elements near cutout   Y-axis - max.stress (N/mm2)

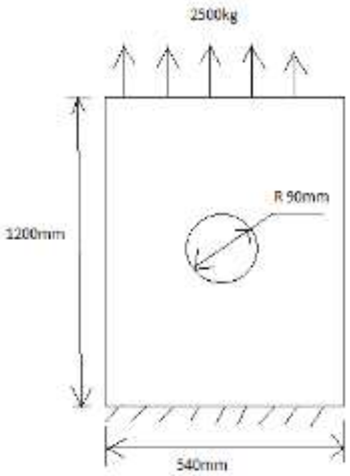
6. RESULT OF CONVERGENCE GRAPH

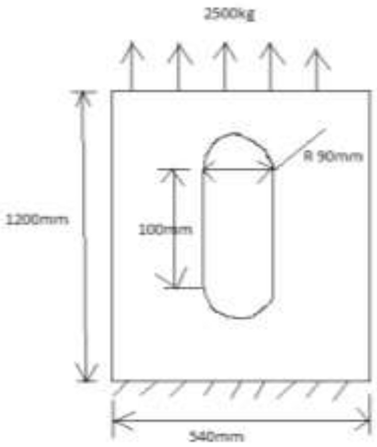
From the convergence graph, for what element number the curve starts converging was identified. We can predict that the convergence element number will give us the exact value which we want. So I was selected 96 is the number of element which is to be maintained near my model cutout, because it initiates the curve to be converge.

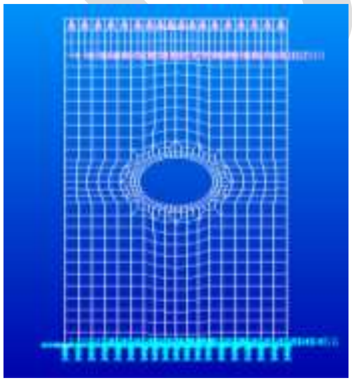
7. COMPARISON STUDY

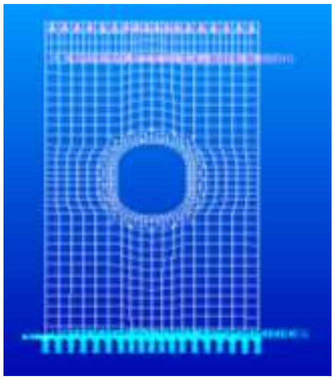
Circular cutout

capsule cutout









8. ANALYSIS OF BOTH CUTOUTS

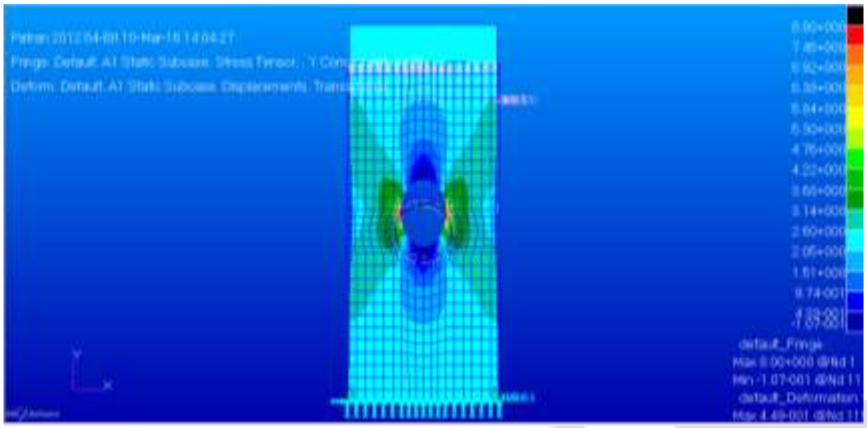


Fig.8.1 (a) Analysis of circular cutout

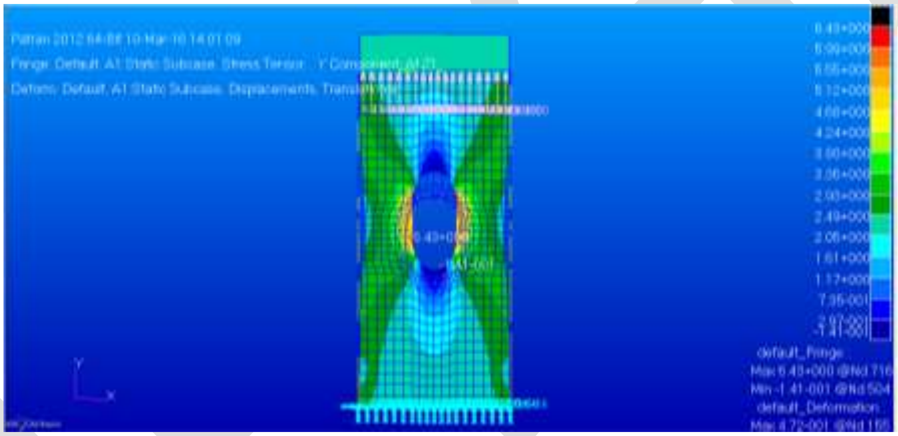


Fig.8.1 (b) Analysis of capsule cutout

9. ANALYSIS OF BOTH CUTOUTS

From the analysis, maximum stress for circular cutout is 8 N/mm2 and maximum stress for capsule cutout is 6.43 N/mm2. This shows the stress concentration in the capsule cutout is less than the circular cutout. So I selected the circular cutout for my model.

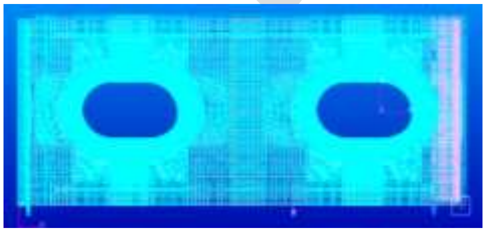


Fig.9.1 Loads and boundary conditions

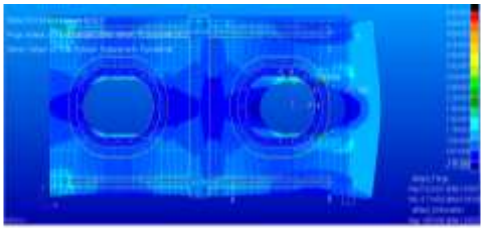


Fig.9.2 Analysis

ANALYSIS OF A MODIFIED MODEL

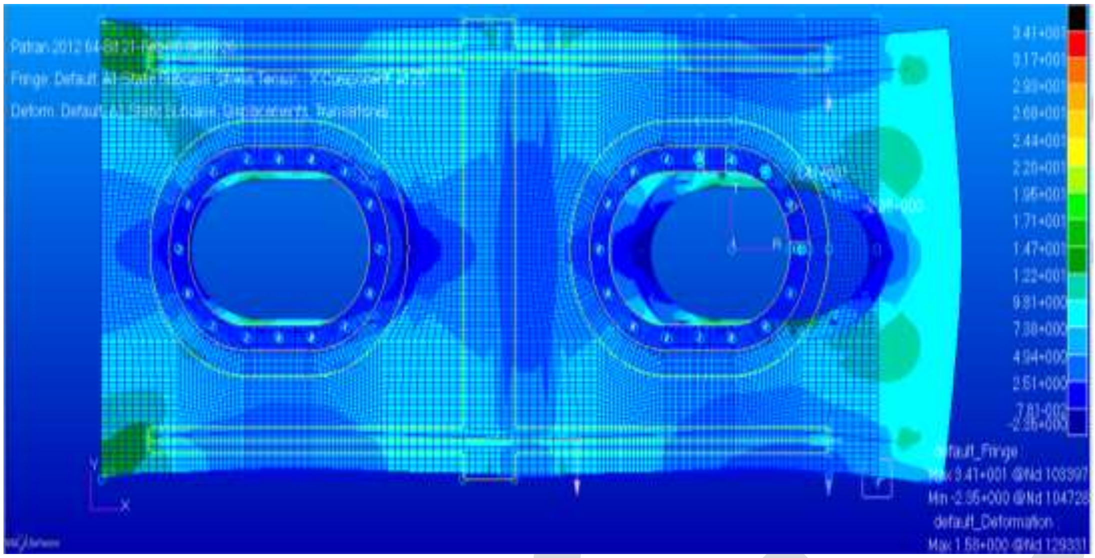


Fig.9.3 Analysis of a modified model

10. RESULTS AND DISCUSSIONS

The stress contour indicates a maximum stress of 34.1N/mm2 at fuel access cutout of wing bottom skin as shown in the above figure. The maximum stress value obtained is within the yield strength of the material. The point of maximum stress is the possible location of crack initiation in the structure due to fatigue loading.

11. CONCLUSIONS

Stress analysis of the fuel access cutout of wing bottom skin is carried out and maximum tensile stress. FEM approach is followed for the stress analysis of the fuel access cutout of wing bottom skin .A validation for FEM approach is carried out by considering a plate with a circular hole. Maximum tensile stress of 34.1N/mm2 is observed in the fuel access cutout of wing bottom skin. Several iterations are carried out to obtain a mesh independent value for the maximum stress. A fatigue crack normally initiates from the location maximum tensile stress in the structure.

REFERENCES:

[1] Sartaj Patel, Mahesha.k, Harish E.R.M, ‘Stress analysis of fuel access cutout of the bottom kin of a transport aircraft.
[2] Sukeerth D A, Thammaiah Gowda, ‘Damage tolerance evaluation for wing structure with large cutout’
[3] Anish nair, K.M Narkar, ‘Structural integrity evaluation of the wing structure at high stress concentration regions’.
[4] Ramesh kumar, S.R.Balakrishnan, S.Balaji, ‘Design of an aircraft wing structure for static analysis and fatigue life prediction’ attachment.
[5] Madhura B M, N.G.S Udupa, Rajanna S, ‘Damage tolerance evaluation of wing in presence of large landing gear cutout through stress analysis using FEM’.