

Optimization of a Fuselage- Stiffened Panel through Stress Analysis

Vivek Hingu¹, Rohit Singh², Vipin Yadav³, Sowmya Gundlathoti⁴, Mansha Kumari⁵

U.G. Student, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India¹

U.G. Student, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India²

U.G. Student, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India³

Assistant Professor, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India⁴

Assistant Professor, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India⁵

ABSTRACT: Aircraft is a flying machine. It is a complex engineering structure. The safety of the structure and the weight of the structure are the two important aspects to be kept in mind while designing the aircraft structure. It is a challenge in front of the aircraft structural designer to bring out a safest structure with minimum weight.

A Finite Element Method (FEM) is used to get estimated solution to a problems ruled by elliptic partial differential equation (PDE) equations. The FEM transforms the elliptic PDE into algebraic equations set, that are simple to solve. The initial value problems which has hyperbolic or parabolic differential equation and initial conditions not completely solved by the FEM. The hyperbolic or parabolic differential equations contain one independent variables as time. The pre-processing task includes building geometric model from CATIA solid model and extracting geometry, building the finite element model, giving properties, setting boundary and loading conditions and finally, assembling elements into a connected structure. This work deals with the buckling analysis of the fuselage stiffened panel structure, on the basis of different cross-sectional view. The Buckling Analysis was done with the NASTRAN & PATRAN software. From the initial cross-section of L shape, it was found out that some modification is needed on the cross-section of the fuselage members, to make it as a safe design. For that, gradually various cross-sections were considered and for different cross-sections, the analysis was carried out and finally obtained an optimum value of Buckling Factor. In the present work, stress analysis of stiffened panel for aircraft fuselage is performed for aluminium alloy plates with is grid and orthogrid using FEM. In the buckling analysis of panel, the linear analysis is carried out under the specified loading and boundary condition and result are extended to the buckling analysis. A linear static analysis will be carried out on a part of fuselage structure with scattered inertia load. The panels with extreme compression load are recognized for buckling analysis. From static analysis, compression load acting on each panel can be extracted and buckling factor is discovered. MSC PATRAN and MSC NASTRAN are used for this study. From the value of Buckling Factor for both Global and Local analysis of fuselage with different cross-sections, it is clear that the design of the fuselage stiffened panel with „L“ and „Z“ cross-section is safe one.

KEYWORDS: Transport aircraft, fuselage, stiffened panel, finite element method, buckling analysis.

I. INTRODUCTION

An aircraft is a complex structure. Each component has one or many functions and designed to guarantee that it can carry out those functions safely and without damaging Fuselage is a component which houses the passengers and cargo in the case of a civil transport aircraft. Fuselage will experience mainly the inertia and pressurization loads (1). Depending on the mass distribution of the fuselage structure the inertia forces will vary along the length of the fuselage. The inertia force distribution makes the fuselage to bend downward about wing axis. This bending of the fuselage will create tension and compression in the upper & lower fuselage stiffened panels respectively. The compression in the lower stiffened panels leads to buckling of the lower panels. The fuselage should carry the payload, and is the main body to which all parts are connected. It must be able to resist bending moments (caused by weight and lift from the tail), torsion loads (caused by fin and rudder) and cabin pressurization. The structural strength and stiffness of the fuselage must be high enough to withstand these loads at the same time; the structural weight must be kept to a minimum. In transport aircraft, the majority of the fuselage is cylindrical or near cylindrical, with tapered nose and tail sections. The basic objectives of the work include evaluation of load carrying capacity of the structure in

presence of crack through the stress analysis of the stiffened panel with and without crack. To ensure the structural integrity of the fuselage through static and buckling analysis using air load on the fuselage. Stress analysis will be carried out to identify panels with maximum compression as critical panels; buckling analysis will be carried out on the critical panel which has maximum compression.

1. Fuselage

The fuselage is an aircraft's main body section. It holds crew, passengers, and cargo. In single-engine aircraft, it will usually contain an engine, as well, although in some amphibious aircraft the single engine is mounted on a pylon attached to the fuselage, which in turn is used as a floating hull. The fuselage also serves to position control and stabilization surfaces in specific relationships to lifting surfaces, which is required for aircraft stability and manoeuvrability.

1.1 Types of fuselage

- Truss structure
- Monocoque structure
- Semi-monocoque structure

1.1.1. Truss structure

This type of structure is still in use in many lightweight aircraft using welded steel tube trusses. A box truss fuselage structure can also be built out of wood often covered with plywood. Simple box structures may be rounded by the addition of supported lightweight stringers, allowing the fabric covering to form a more aerodynamic shape, or one more pleasing to the eye.

1.1.2 Monocoque shell

A monocoque fuselage has its skin holding the skeleton structure together while the semi-monocoque has both the skin and the skeleton holding together. A monocoque design would also have a safety risk involved if the skin was damaged because it is the load-bearing structure.

1.1.3 Semi-monocoque

The term semi-monocoque refers to a stressed shell structure that is similar to a true monocoque, but which derives at least some of its strength from conventional reinforcement. Semi-monocoque construction is used for, among other things, aircraft fuselages, car bodies and motorcycle frames.

1.2 Compare between monocoque and semi monocoque structure

This idea was by Eugene Reconnect. Rather than mounting panels onto a solid load-bearing skeleton (like old time) we can let the airplane's skin carry the load. This by forming the fuselage out of multiple layers of wood. The layers were glued together with their grains running in different directions to strengthen the skin so much that no internal skeleton was needed! He called the new technique monocoque construction, or single-shell construction. Monocoque construction works fine with tiny airplanes, but there is a point at which the weight of the monocoque fuselage is greater than the weight of an internal skeleton. Furthermore, monocoque construction is unforgiving of even the slightest structural failure: think of how easily aluminium can be crushed once a single dimple forms in its skin and you'll see the inherent problem.

Why do mostly commercial aircraft use semi-monocoque fuselage in construction?

The semi-monocoque construction uses the skin of the aircraft as the primary load (of pressurization) bearing structure. This reduces the weight of the aircraft hence making it very economical to fly. This Type of construction uses Formers and Stringers to help the skin bear the pressurization loads, hence making it effective other than just being light weight. Commercial operations are all about how we convert Investment into maximum profit. In aviation this is done by saving as much fuel as possible. This will only be possible if the aircraft is lightweight so that it can carry other payload to generate profits.

II. METHODOLOGY

Al 2124-T851 is used in current Fuselage section due to high strength and fatigue resistance properties. In a structural analysis problem, generally trying to determine the response of a model to physical loading, or specific structural behaviour such as frequency response or buckling. Some of the loads and boundary conditions may work with include force, pressure, velocity, inertial loads, displacement, temperature, and contact.

Introduction to CAD 3D modelling

3D modelling or CAD (Computer Aided Design) allows engineers and designers to build realistic computer models of parts and assemblies. These models can be then 3D Printed or CNC machined as well as used to run complex simulations.

Modelling of Stiffened Panel using CATIA

Stress analysis of the fuselage segment followed by stiffened panel analysis Stress intensity factor calculations using fracture mechanics principles. The three dimensional model of the stiffened panel is generated using computer aided Three dimensional interactive application (CATIA) V5 package as per the given Specifications. The model is meshed using PATRAN. 2Dimensional stress analysis is carried out using MSC Nastran.

To the meshed component material properties are described. Also the component is subjected to loading and applying boundary conditions to determine the maximum stress in the component. The stress distribution is obtained from static analysis using NASTRAN. The panel with maximum compression is considered as critical panel for the buckling analysis for three different skin thicknesses. The linear buckling analysis is carried out on panel with maximum compression. The buckling analysis is carried out on each panel of three different skin thicknesses for the segment of the fuselage Which have maximum compression?

Calculation of the critical buckling load using analytical approach (using EULER's buckling formula for thin sheets) Results are substantiated with finite element method i.e. results obtained from analytical results are compared with **FEM results**.

Specification of Transport Aircraft Fuselage

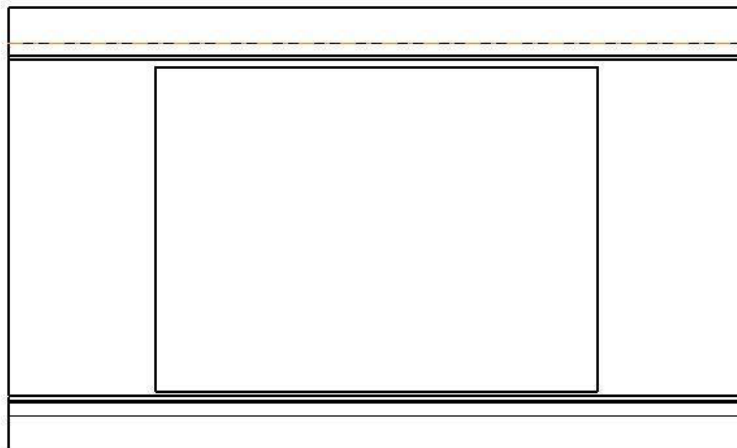


Fig.1: StiffenedPanel

III. FEM DESIGN

The finite element method (FEM) has now become a very important tool of engineering analysis. Its versatility is reflected in its popularity among engineers and designers belonging to nearly all the engineering disciplines. Whether a civil engineer designing bridges, dams or a mechanical engineers designing auto engines, rolling mills, machine tools or an aerospace engineer interested in the analysis of dynamics of an aero plane or temperature rise in the heat shield of a space shuttle or a metallurgist concerned about the influence of a rolling operation on the microstructure of a rolled product or an electrical engineer interested in analysis of the electromagnetic field in electrical machinery-all find the finite element method handy and useful. It is not that these problems remained unproved before the finite element method came into vogue; rather this method has become popular due to its relative simplicity of approach and accuracy of results. Traditional methods of engineering analysis, while attempting to solve an engineering problem mathematically, always try for simplified formulation in order to overcome the various complexities involved in exact mathematical formulation. In the modern technological Environment the conventional methodology of design cannot compete with the modern trends of Computer Aided Engineering (CAE) techniques. The constant search for new innovative design in the engineering field is a common trend. To build highly optimized product, this is the basic requirement of today for survival in the global market. All round

efforts were put forward in this direction. Software professional and technologists have developed various design packages.

For analysis NASTRAN and PATRAN software are used. This software's were well known as Finite Element Analysis. The finite element method is a numerical technique for solving engineering problems. It is most powerful analysis tool used to solve simple to complicated problems. The pre-processing stage involves the preparation of nodal co-ordinates & its connectivity, meshing the model, load & boundary conditions and material information for finite element models carried in MSC PATRAN described in the figure above. The processing stage involves stiffness generation, modification and solution of equations resulting in the evaluation of nodal variables, run in MSC NASTRAN. The post-processing stage deals with the presentation of results, typically the deformed configurations, elemental stresses and forces etc.

Phases in FEM

Pre- processing, in which the analyst develops a finite element mesh to divide the given geometry into subdomains for mathematical analysis, and applies material properties and boundary conditions.

Solution, during which the program derives the governing matrix equations from the model and solves the primary quantities.

Post- processing, in which the analyst checks the validity of the solution, examines the values of the primary quantities, and derives and examines additional quantities.

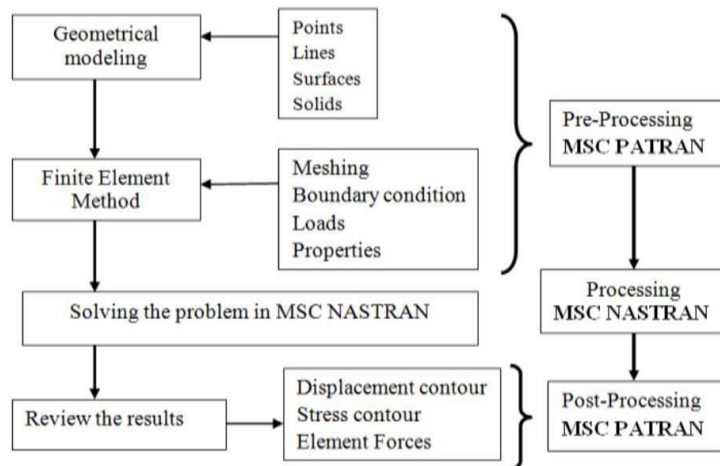


Fig.2: Steps Involved in Finite Element Analysis

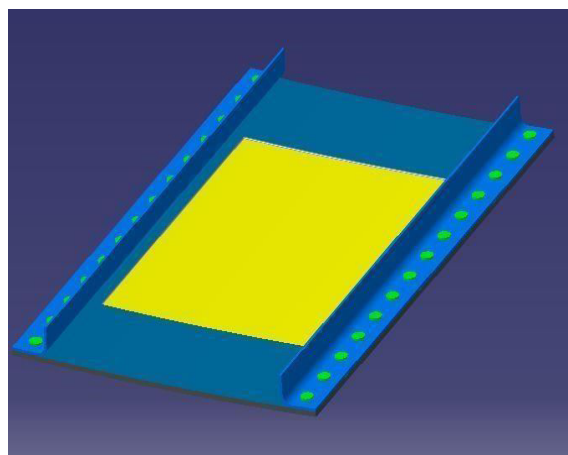


Fig.3: Stiffened panel

IV. CONCLUSION

The stiffened panel for the fuselage has been designed and analyzed for stress using CATIA and MSC NASTRAN PATRAN software. The level of deformations and frequencies observed are close to the theoretical values. The maximum deformation of the stiffened panel under given loading and boundary conditions is. The experimental results

of stiffened panels for buckling response under compression loading were compared with the numerical results obtained using linear and nonlinear finite element analysis

REFERENCES

- [1] D.J. Dawe, S. Wang, "Postbuckling analysis of composite laminated panels", American Institute of Aeronautics and Astronautics Journal, 38 (2000).
- [2] J.W. Hutchinson, W.T. Koiter, "Post buckling theory", Applied Mechanics Reviews, 23 (1970), pp. 1353-1366.
- [3] T.M. Roberts, Z.G. Azizian, "Strength of perforated plates subjected to in-plane loading", Thin-Walled Structures, 2 (2) (1984), pp. 153-164.
- [4] Yadav, Shatjit, and M. Ramachandran. "Analysis of Optimized Wind Turbine Failure Using Computational Fluid Dynamics." Materials Today: Proceedings 4, no. 2 (2017): 1788-1793.
- [5] W.H. Wittrick, F.W. Williams, "Buckling and vibration of anisotropic or isotropic plate assemblies under combined loadings", International Journal of Mechanical Sciences, 16 (4) (1974), pp. 209239
- [6] Ramanan.G, Neela Rajan.R, Diju samuel.G, Edwin Raja Dhas.J, "Multiple Response Characteristics Optimization of WEDM Pa Rameter.
- [7] D. Quinn, A. Murphy, W. McEwan, F. Lemaitre, "Stiffened panel stability behaviour and performance gains with plate prismatic substiffening", Thin-Walled Structures, 47 (2009), pp. 1457-1468.
- [8] C. Bisagni and P. Cordisco, "Testing of Stiffened Composite Cylindrical Shells in the Post Buckling Range until Failure", American Institute of Aeronautics and Astronautics Journal, 42, (2004), 1806-1817.
- [9] Ramanan.G, Edwin Raja Dhas.J and Ramachandran.M, "Optimization of material removal rate and surface roughness for wire electric discharge machining of AA7075 composites using grey relational analysis", International Journal of Vehicle Structures and Systems, Vol.9, No.5 (2017), pp. 309-312.
- [10] Rajesh Prabha.N, Edwin Raja Dhas.J and Ramanan.G., "Finite element structural analysis of connecting rod of AA7075-TiC composite using ANSYS", International Journal of Mechanical Engineering and Technology, Vol.8, 2017, pp. 1102-1110.
- [11] Ragavendran, U., Viral Mehta, Vishal Fegade, and M. Ramachandran. "Dynamic Analysis of Single FoldSymmetric Composite Laminates." International Journal of Civil Engineering and Technology 8, no. 11 (2017): 536545.