IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

Design And Modelling Of Stiffened Panel For Automotive Applications

SHANKRAPPA[1], ANANDA KOPPALADA[2], RAGHAVENDRA PATIL[3]

[1,2,3] Lecturer in Department of Mechanical Engineering
[1,2] Government Polytechnic, Karatagi, Karnataka, India
[3] Government Polytechnic, Kampli, Karnataka, India

Abstract: The primary tasks of an aircraft's structure are to transmit and resist applied loads; to give an aerodynamic form; and to protect passengers, payload systems, and other components from the ambient conditions encountered during flight. Most airplanes are designed with thin shell structures, with the outside surface or skin supported by longitudinal stiffening members and transverse frames to resist bending, compressive, and torsional stresses without buckling. Such structures are known as semi-monocoques, whereas thin shells that rely solely on their skins for load resistance are known as monocoques.

Index Terms – Catia V 5, FEM, Nastran.

I INTRODUCTION:

The finite element method (FEM) is a numerical tool for solving problems with partial differential equations or functional minimization. A domain of interest is represented as a set of finite elements. Approximating functions in finite elements are defined in terms of the nodal values of the physical field being sought. A continuous physical problem is converted into a discontinuous finite element problem with undetermined nodal values. A linear issue requires the solution of a set of linear algebraic equations. Values within finite elements can be recovered using nodal values.

Two elements of the FEM deserve to be mentioned:

1) The piece-wise approximation of physical fields on finite elements provides good precision even with simple approximating functions.

2) Locality of approximation results in sparse equation systems for a dubious problem. This is useful for solving issues with a large number of nodal unknowns.

To summarize in general terms how the finite element method works we list main steps of the finite element solution procedure below.

1. Discretize the continuum: The first step is to divide a solution region into finite elements. The finite element mesh is typically generated by a preprocessor program, which is MSC Patran in our case. The description of mesh consists of several arrays, main of which are nodal coordinates and element connectivity.

2. Select interpolation functions: Interpolation functions are used to interpolate the field variables over the element. Often, polynomials are selected as interpolation functions. The degree of the polynomial depends on the number of nodes assigned to the element.

3. Find the element properties: The matrix equation for the finite element should be established which relates the nodal values of the unknown function to other parameters. For this task different approaches can be used; the most convenient are: the variation AL approach and the Galerkin method.

4. Assemble the element equations: To find the global equation system for the whole solution region, we must assemble all the element equations. In other words we must combine local element equations for all elements used for discretization. Before solution, boundary conditions (which are not accounted in element equations) should be imposed.

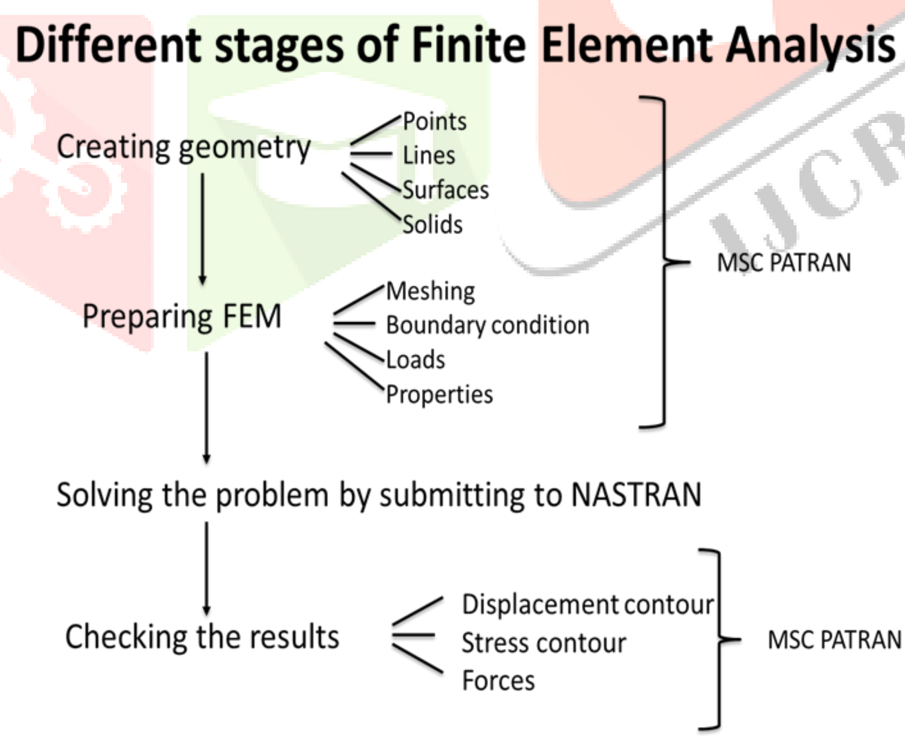
5. Solve the global equation system: The finite element global equation system is typically sparse and symmetric. Direct and iterative methods can be used for solution. The nodal values of the sought function are produced as a result of the solution.

6. Compute additional results: In many cases, we need to calculate additional parameters. For example, in mechanical problems strains and stresses are of interest in addition to displacements, which are obtained after solution of the global equation system. So, these values of displacements can be used to find other parameters.

The different stages of FEA:

The software used for the analysis of the Landing gear lug attachment joint in an airframe is MSC Patran & MSC Nastran. The stages involved in FEM are shown in the figure below,

Different stages of Finite Element Analysis



CATIA V5:

CATIA V5 is mechanical design software, addressing advanced process centric design requirement of the mechanical industry. This tool makes it possible for mechanical designers to quickly sketch ideas, experiment with features and dimensions, and produce models and detailed drawings. The following commands are commonly used in geometric modeling. One can create geometrical drawing using 2D sketched geometry only, without reference to existing models or assemblies.

This sketched geometry can be controlled by relations (collinear, parallel, tangent, and so on), as well as parametric dimensions. Extrude, using this option one can extrude base features and other features using 2D sketch. Revolve command can creates a feature that adds or removes material by revolving one or more profiles around a centerline. Pattern command can create a linear pattern, a circular pattern, a curve driven pattern, or use sketch points or table coordinates to create the pattern. Mirror, command copies the selected features or all features, mirroring them about the selected plane or face. Circular pattern command used to create multiple instances of one or more features, which we can space uniformly around an axis.

Fillet and Chamfer command can be used to create fillet all edges of a face, selected sets of faces, selected edges, or edge loops and beveled feature on selected edges or a vertex. Cut, option is used to trim features and 3Dmodel with respect to a defined plane. In the present work geometric models was created by using all these commands.

MSC PATRAN

Developing a finite element model manually is a time consuming, tedious and error prone activity making sense of the large stake of finite element computer output is also a considerable challenge.

A finite element pre and post processors (such as MSC/PATRAN) is a graphic based software package primarily designed to aid in the development of Finite Element Model (Pre processing) and to aid the display and interpretation of analysis results (Post processing). MSC/ PATRAN software is a mechanical computer aided engineering tool created for design engineers.

Utilizing integrated automatic technologies MSC/PATRAN enables design engineers to build and modify solid models of computer parts and predicts their behavior through design optimization. In addition preprocessing software helps the analyst modify the model if the result shows that changes and subsequent reanalysis are required.

Some pre processors able to import geometric data from solid modeling or computer aided design manufacturing (CAD/CAM) software to be used as a basis for the finite element model.

Components of MSC PATRAN:

Pre-Processor: Preprocessing involves the preparation of data such as nodal co- ordinates, connectivity boundary conditions, loading and material information. The pre –processor takes minimum input for the user, creates the finite element mesh and other data required for analysis.

Solver/processor: The processing stage involves stiffness generation, modification and solution of equations resulting in the evaluation of nodal variables.

Post Processing: The post processing stage deals with the presentation of results. Typically the Deformed configurations, mode shape, displayed at this stage. Graphical post- Processing of results helps to receive the physical consequences of the analysis

III.LITERATURE REVIEW:

Thomas P. Rich: Developed a general equation for predicting structural failure of two-dimensional cracked components, in which the component's geometrical properties influence the crack's stress intensity factor. The general equation is used to create a new fracture diagram for an evenly stressed sheet with a crack that is limited by two stiffening components fastened to the sheet. Using an example, it is demonstrated that the shape of the fracture diagram, and hence the panel's fracture behavior, is determined by the spacing of the stiffening elements, rivet placements, and the relative stiffness of the cracked sheet and stiffeners.

Pir M. Toor: focuses on building a fail-safe fuselage construction that takes into account circumferential cracks caused by vertical fuselage bending loads. The investigation of these types of cracks is complicated, initially because of complex structural configuration (frames, skin Longerons, and crack stopper straps), and then because of the influence of the shell's curvature. He also discovered that Longeron (stringers) are more effective in preventing circumferential cracks.

IV: INTRODUCTION TO MSC NASTRAN:

Features of MSC NASTRAN

1.Linear static analysis.
2.Static analysis with geometric and material non-linearity.
3.Transient analysis with geometric and material non-linearity.
4.Normal mode sand buckling analysis.
5.Direct and model vibration analysis (including response spectrum analysis)

Linear static analysis

1.Multi load case and load case combination
2.Equivalent stress concentration
3.Internal shear and edge effects in composites

Material properties

1.Isotropic or orthographic
2.Temperature dependent
3.Directional tensile, compressive and shear failure stress for composite

Loading

1.Point force, momentum on nodes or range of nodes in local or global coordinate
2.System
3.Pressure load
4.Linear, angular acceleration, angular velocities

Kinematics constraints

1.Specified Nodal Displacement
2.Coupled Displacement
3.Multiple constrains

Output

1.Displacement and stress at elements and nodes
2.Reaction forces
3.Element strain energy and internal forces

V. RESULT AND DISCUSSION:

Complete Finite element model of the stiffened panel

Finite element meshing is carried out for all the components of the stiffened panel such that there is a node present at the point where riveting is to be done and fine meshing is done at the critical sections where stresses are expected to be more.

The complete Finite element model of the stiffened panel as show in figure 1

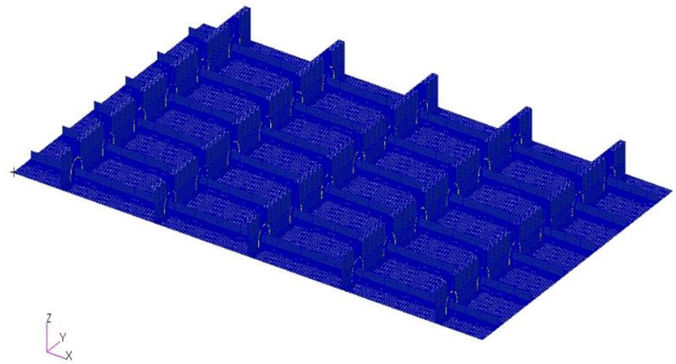


Fig 1: Finite element model of the stiffened panel

Finite element meshing is carried out for all the components of the stiffened panel such that there is a node present at the point where riveting is to be done and fine meshing is done at the critical sections where stresses are expected to be more.

The following figures show the details about the finite element mesh generated on each part of the structure using MSC PATRAN.

skin:

The fig 2 shows the finite element mesh on skin. The skin houses rest of the components like bulkheads, longerons, tear strap. The mesh was carefully generated such that there is a node present at the point where riveting is to be done. The fig 3 shows the rivets that are placed on the skin to hold the frames, tear strap and longeron together.

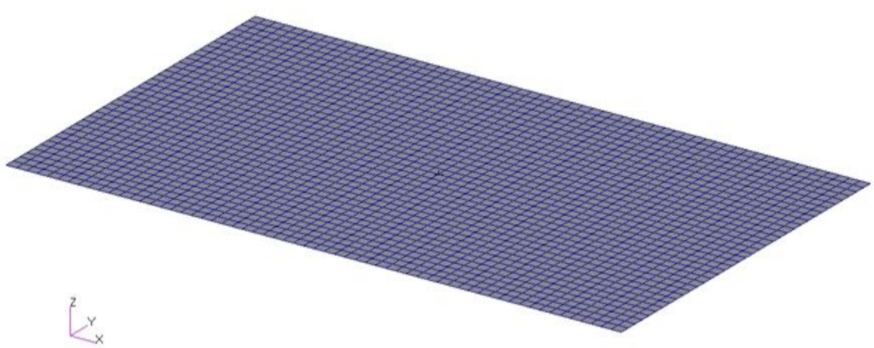


Fig 2: Finite element Mesh on skin

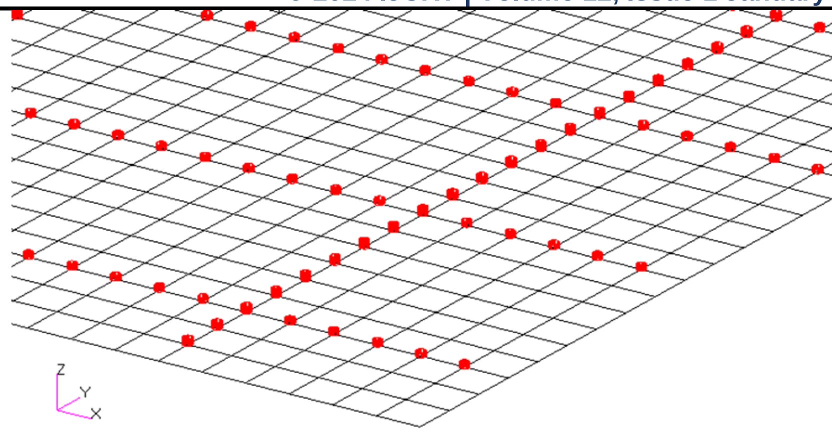


Fig 3 :Close up view of mesh on the skin with beam elements as rivets

Riveting is carried out by selecting the node on the skin and the corresponding node on the other component and created a beam element between them. So, the riveting process is completed in his manner. Longeron (Stringers)The Longeron is also known as stringer, in our case, it is of L cross-section, and it runs from tip to tip of the fuselage in the longitudinal direction. The Finite element mesh for stringer shown in fig 4.

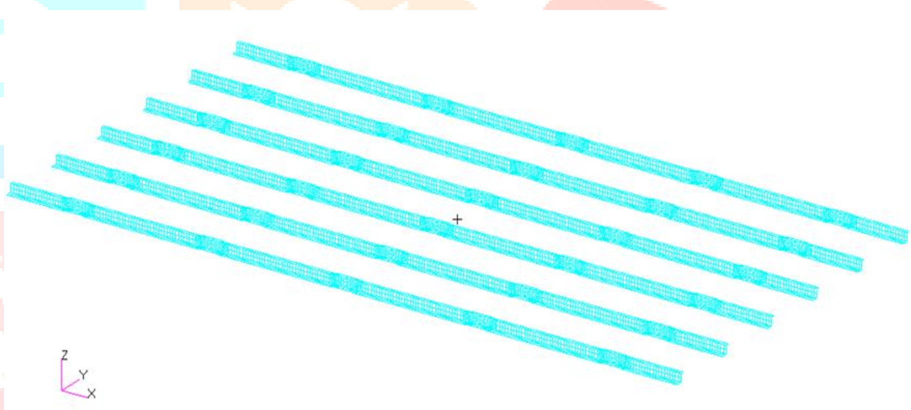


Fig 4: Finite element mesh on Longeron

These stringers are placed on top of the Skin and Tear strap, and then riveted into its position. The tear strap runs under the stringer and enough space is provided for this purpose by bending the longeron at calculated distances as shown in fig 4.

Bulkhead:

Bulkhead is also known as frame. Bulkhead is a stiffening member in circumferential direction in the fuselage structure. There are five bulkheads in this stiffened panel. Finite element modeling of the bulkheads are shown in fig 5.

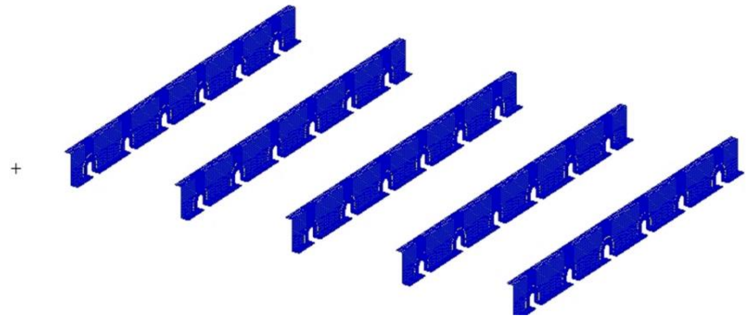


Fig 5 :Bulkheads

VI.CONCLUSION:

A stiffened panel which is genetic structural element of the fuselage structure is evaluated analytically for its crack arrest capability.The internal pressure is one of the main loads that the fuselage needs to hold. In the current project also pressurization load case is considered for the analysis.Finite element analysis (FEA) approach is used for structural analysis of the stiffened panel.

REFERENCE:

[1] Thomas P. Rich, Mansoor M. Ghassem, David J. Cartwright, “Fracture diagram for crack stiffened panel”, Engineering Fracture Mechanics, Volume 21, Issue 5, 1985, Pages 1005-1017

[2] Amy L. Cowan “Crack path bifurcation at a tear strap in a pressurized stiffened cylindrical Shell” in August 24, 1999

[3]laxmi narasia.3rd edition. bs publication 2010 …

[4] www.mscsoftware.com