STRUCTURAL AND MODAL ANALYSIS OF FUSELAGE

[1]Y Santosh, [2]Prashanth Bhatti

[1]M.Tech. Student, [2]Assistant Professor
Department of Mechanical Engineering, CAD/CAM.
MLR Institute of Technology, Hyderabad, India.

***Abstract*- The conceptual design of fuselage structure for an aircraft by using CAD software (CatiaV5) as the design tool. Specific size and performance, the number of competing designs and the commonality of features with existing aircraft are factors need to be considered in the design process. This conceptual design develops the first general size and configuration for aircraft fuselage structure.**

The model of the fuselage structure is then undergoing engineering simulation programmed which is based on the finite element method. Structural and modal analysis were performed using stringers and bulk head of a fuselage. A cylindrical shell is assumed as the design space and aerospace standard pay loads were applied on the fuselage with wing attachments as constraints. Then topological optimization is done using Finite Element (FE) based software (Hypermesh). This optimization results in the structural concept design which satisfies all the design constraints using minimum material.

***Keywords*: Fuselage analysis Catia**

I. INTRODUCTION

The major challenge in today's ground vehicle industry is to overcome the increasing demands of higher performance, lower weight, and longer life of components, all this at a reasonable cost and in a short period of time. The fuselage structure basically consists of skin panels connected directly to frames and stringers for longitudinal splices. The stiffening members have to cross each other at 90˚. Therefore a provision is made to pass the longitudinal stiffeners through the bulkheads which are in the circumferential direction. These cut-outs in the bulkheads will act as stress raisers due to hoop tension in the bulkheads. Cabin pressure results in radial growth of the skin and this radial growth is resisted by frames and stringers giving local bending along the fastener lines. Fuselage skin panels are curved and these panels are under biaxial tension loading due to cabin pressure. Many modern aircraft are being designed to operate at high altitudes, taking advantage of that environment. In order to fly at higher altitudes, the aircraft must be pressurized. As the altitude of the aircraft changes the cabin must be pressurized or depressurized respectively to maintain the cabin pressure constant, this pressurization and depressurization leads to the cyclic loading on the fuselage. Therefore this internal pressurization is considered to be critical load cases during the design and development of the aircraft.

It is estimated that fatigue failure is multi-stage process that begins with crack initiation, propagating with continued fatigue is responsible for 85% to 90% of all structural failures.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), torsional loads (caused by fin and rubber) 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 semi-monocoque construction, which is virtually standard in all modern aircraft, consists of a stressed skin with added stringers to prevent buckling, attached to hoop-shaped frames.

1. Topology Optimization methods

Several methods have been used for implementing topology optimization to determine material distribution on a given design domain. Some of the popular methods are:

- Ground Structure Approach
- Solid Isotropic Material with Penalization (SIMP)
- Homogenization
- Level Set Method
- Evolutionary Structural Optimization (ESO)
- Genetic Algorithms

Among these methods SIMP method is chosen for implementation in this project. SIMP is the most popular method in CAE packages.

1.1. Solid Isotropic Material with Penalization (SIMP) Approach

The idea of parameterizing the design domain rather than solving a discrete on-off problem in the field of topology optimization was first documented by Bendsoe in the late 1980's. Consequently two methods, namely, SIMP and homogenization, received much attention in the early 1990's. The SIMP method, also called the power law method, the direct approach, or the artificial density method, works by keeping a fixed finite element discretization and associating with each finite element a density function $\rho(x)$ whose values lie between 0 and 1. A zero denotes a void and a 1 denotes a solid. The material properties of a particular element are a function of its density so that element densities can be used as design variables to adjust the performance of the design. Thus, if the solid (ρ(x) =1) material property, say, Young‘s modulus in structural optimization is denoted by Esthen the young‘s modulus of elements with intermediate densities is given byEi = Es* ρi(x) where the subscript ‘i‘, denotes a particular element.The penalization technique used is the "power law representation of elasticity properties," [9] which can be expressed for any solid 3-D or 2-D element as following equation.

$$\underline{K}(\rho) = \rho^{p} K$$

Where,
$\underline{K}$: is penalized stiffness matrix of an element
K: is the real stiffness matrix of an element
ρ: is the density
p: is the penalization factor which is always greater than 1.
Thus, the fundamental equation that characterizes a SIMP approach is given by the following Equation.

$$E_i = \rho_i(x)^p E_s \quad ; \quad Vol = \int_{R^3} \rho(x)dx$$

Where Eiis the young‘s modulus of the ‘ith’ element in the design domain, ρiis the artificial density parameter for the ’ith‘element, p is the penalization parameter for intermediate densities (usually, p ≥ 3), Esis the solid material young‘s modulus, Vol, is the total volume from distributed material in the design domain, R3 denotes a three dimensional design region and ‘x’ denotes the coordinate points in space for the centroid of a material element.
The SIMP approach has the following important advantages:
1) It uses a simple parameterization technique that is very easy to implement.
2) It has been extensively studied and applied to problems with complicated design conditions.
3) It uses only one design parameter, the density, for each element and thus requires less storage space and computational effort.
4) The SIMP method eliminates intermediate densities by raising them to a high power ‘p‘ and thus overcomes both the inaccuracy of the parameterization model at intermediate densities and the difficulty of fabricating those intermediate densities with conventional non-additive manufacturing processes.
As mentioned earlier Altair® OptiStruct® is an industry proven, modern structural analysis solver for linear and non-linear structural problems under static and dynamic loadings. OptiStruct solves topological optimization problems using the density method, i.e., the SIMP method. Moreover any changes required to the model for optimization can be easily incorporated in Optistruct.

II. 3D MODELING OF FUSELAGE

Fuselage is the main body of the aircraft which carries maximum loads, it is an assembly consists of SKIN supporting structureCAD model of the stringer after using pad operation and there are 16 stringers in fuselage assembly to support the structure from heavy loads. 16 stringers are modeled using the circular pattern definition with the angular spacing of 22.5 deg. Rivets are modeled using shaft option with respective to x axis and these rivets are the connectors between skin and the stringers and it has been spread throughout the length of stringers and skin. Final assembly including stringers, frame, skin, rivets and rivet frame is shown in Figure 3.10.

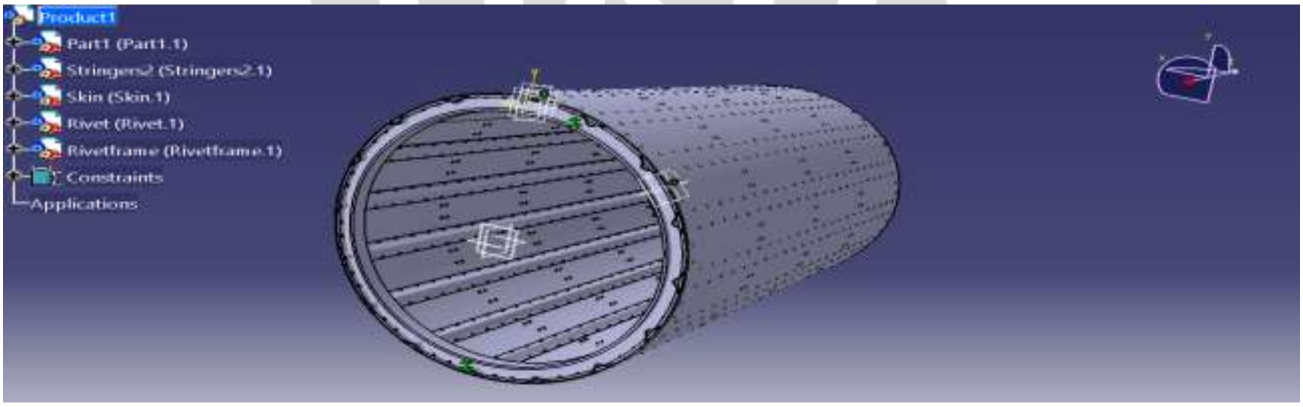

Fig 3.10 Final assembly model of fuselage.

III. STRUCTURAL ANALYSIS

Fuselage Meshing typically starts with the import of given CAD data.Some of the issues do exist because when designers create CAD geometry; their priorities are different from those of analysts trying to use the data.The basic idea of FEA is to make calculations at only limited (Finite) number of points and then interpolate the results for the entire domain (surface or volume). Any continuous object has infinite degrees of freedom and it’s just not possible to solve the problem in this format. Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).Quadratic and triangular elements are used to mesh the fuselage skin, frame, and stringers.

3.1 Rigid Element Connection RBE2

A center node is connected to the outer edge nodes using a rigid element (RBE2). The torque is then applied at the center node. Rivets and rivets frame are too small compared to other parts of the fuselage. So,nodel these 1-D elements is used.

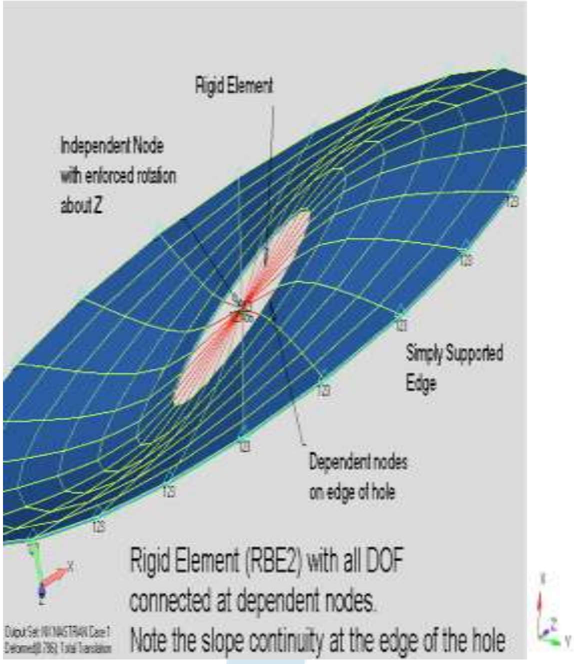

Fig 3.1.1 Representation of RBE2

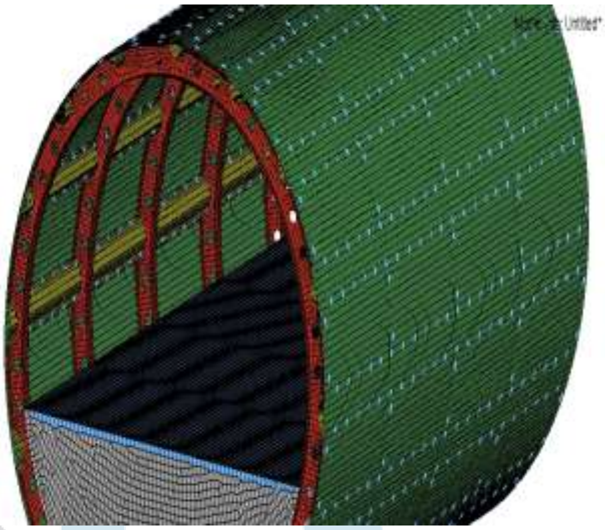

Fig.3.1.2Meshed assembly model of fuselage

For an RBE2 rigid element, the single node is the **"independent node".** It has 6 degrees of freedom, regardless of what your FEA graphical interface looks like. The other node(s) are the dependent nodes. We decide which of their degrees of freedom we wish to connect. Note that it is important to understand the significance of our choices. Typical choices for the connected DOF would be either TX, TY, TZ or all 6 DOF (i.e. translations and rotations).

1. If we have multiple rigid elements, do not have any node(s) as the dependent node(s) of more than one rigid element. **Rigid elements can be nested** (i.e. the dependent node of one rigid element can be the independent node for another rigid element), however you should avoid a full circular dependency.

2. Do not apply constraints to the dependent nodes of a rigid element. If we do, we need to be sure that the DOFs that you constrain are not the same DOF's affected/connected in the element.

3. An important reminder is that rigid elements operate using small displacement theory. For example, if we use an RBE2 to rotate some structure, the initial (infinitesimal) vector of displacement at each dependent node does not change its direction irrespective of the magnitude of rotation at the independent node. In the case of basic non-linear analysis, this condition remains true.

After meshing all the components of the fuselage and it is connected with the help of the connectors, the final meshed assembly model is shown in Figure 3.2

The Quality Index panel calculates a single value, called compound Q.I., which represents the quality of the displayed shell (2D) model. This value is calculated based on criteria entered by the user and can be used to optimize the quality of the 2D mesh. Fig 3.3 shows quality index performed in HyperMesh for fuselage assembly.

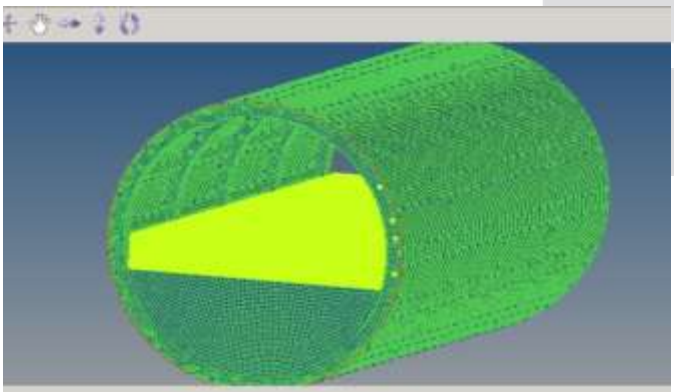

Fig 3.1.3 Quality Index

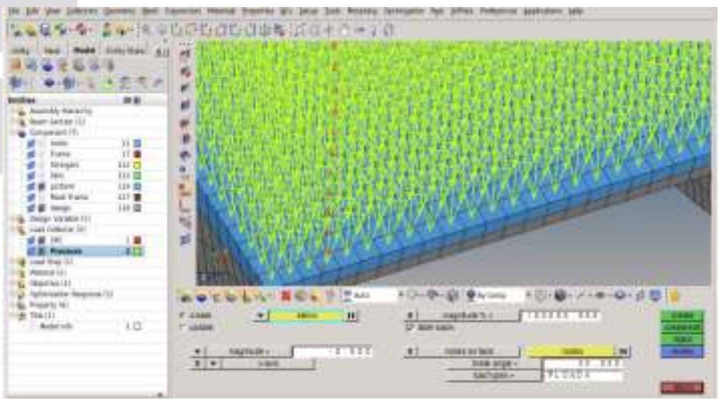

Fig 3.1.4 Pressure loads

3.2 Material Properties

Aluminum 6061-T6, it is widely used in the applications of aerospace engineering and mechanical properties of Al alloy is given in Table 3.2.1

Table 3.2.1 Mechanical properties of Aluminum 6061-T6

Properties	Value
Ultimate Strength	310Mpa
Yield Strength	276Mpa
Modulus of Elasticity	68.9Gpa
Poisson's ratio	0.33
Fatigue Strength	96.5Mpa
Shear Strength	207Mpa

Uses of the Aluminum 6061-T6 material in aircraft assembly other than fuselage are, Aircraft fittings, camera lens mounts, coupling, marines fittings and hardware, electrical fittings and connectors, decorative or misc, hardware, hinge pins, magneto parts, brake pistons, hydraulic pistons, appliance fittings, valves and valves parts, and bike frames.

3.2.1 Loads and boundary conditions

Fuselage is fixed at the both ends and pressure loads are applied on the face of the platform (Design Space) and shown in Figure 3.1.4. Therefore whole body is constrained with all DOF in space. Single Point Constraint (SPC) defines the restricted degrees of freedom.Loads applied are approximately taken as per person 80 kgs, total persons are assumed to be 110.

The length of the fuselage is very large and the number of elements also, so the need of pressure loads is required it distributes equally the total load applied on the each face of the element. Pressure loads is calculated as length multiplied by breath divided by the number of elements in the design space. This SPC and Pressure loads are created in load collectors in the optimization software.

3.3 Static Analysis results of Fuselage

Several results for evaluating stress analysis are available to the user. The counter plots of a specific result over the model are shown in Figs 3.3.1. The Finite Element Model with the boundary conditions is submitted to Optistruct solver for Linear Static Analysis. HyperView is used for post-processing. For checking the elemental stresses and forces at nodes in local region have to check a grid point force balance, elemental forces, grid point stresses and elemental stresses.

3.3.1 Displacement contour increases circumferentially and it is shown by different colors fringes where blue color showing minimum magnitude of displacement while red color showing maximum magnitude of displacement as 0.126.

3.3.2Stress contour of the fuselage structure The Von misses stresses (227.4 Mpa) are below the yield strength of the Aluminum 6061-T6 (276Mpa). Hence it is treated as safe model.

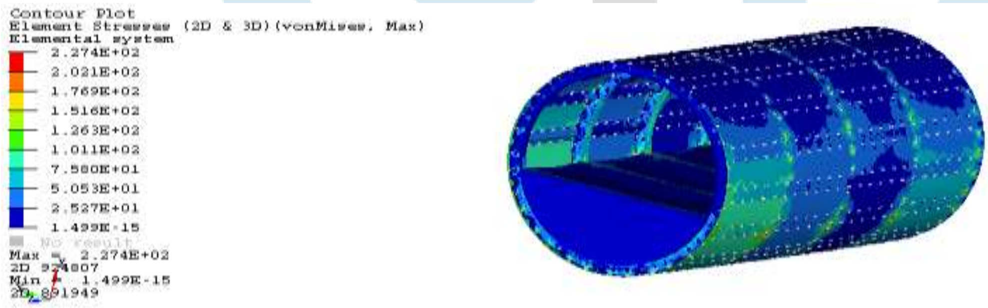

Fig 3.3.1 Elements Stresses (2D & 3D)

3.3.3Stress contour of fuselage result, it is clear that the maximum stress is on the platform and minimum stress is at the frame of the fuselage structure and the direction can be observed from the Fig 3.3.2. The maximum stress locations are the probable locations for crack initiation and close-up view of maximum stress is shown in Fig 3.3.2. Invariably these locations will be at platform in design space. Representation of layered structure is important in identifying critical stress locations, integral representations will miss lead as for as critical locations are concerned.

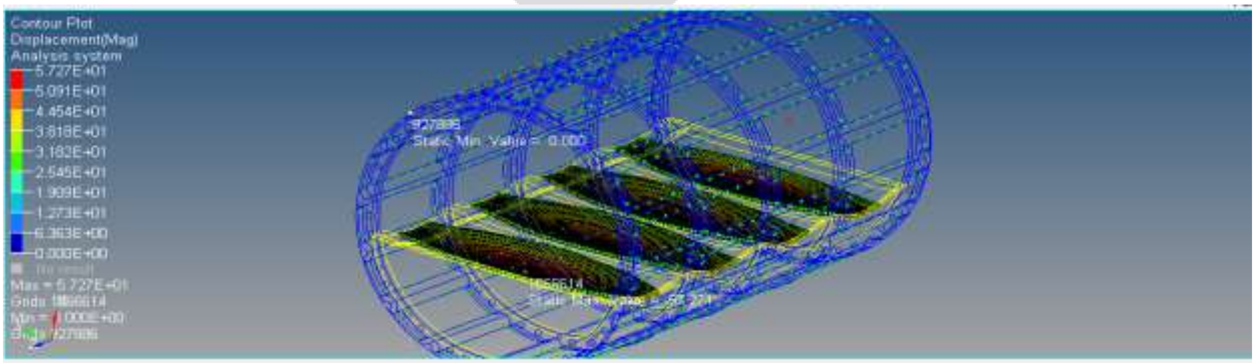

Fig 3.3.2Identifying the critical locations for applied load.

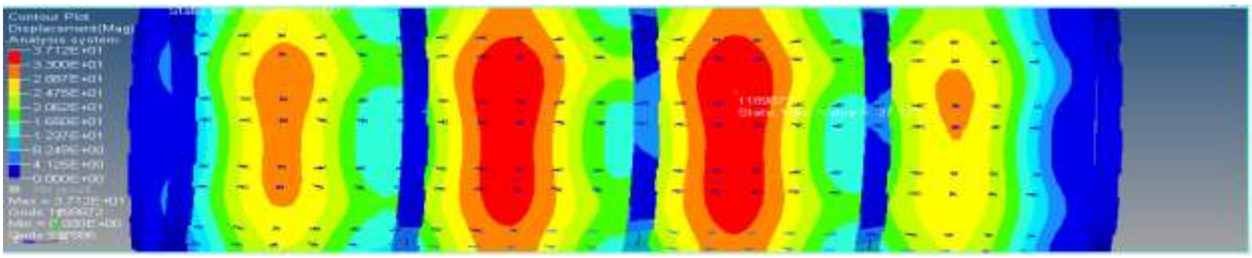

Fig 3.3.3 Close up view of maximum stress location.

IV. Modal Analysis

4.1 Modal Analysis Preparation

Modal analysis to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It also can be a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis. Modal analysis is the field of measuring and analyzing the dynamic response of structures and or fluids during excitation.

4.2 Modal Analysis Results

From the modal analysis, some mode shapes are extracted and are shown in Fig 4.2 (a to g) including different frames.

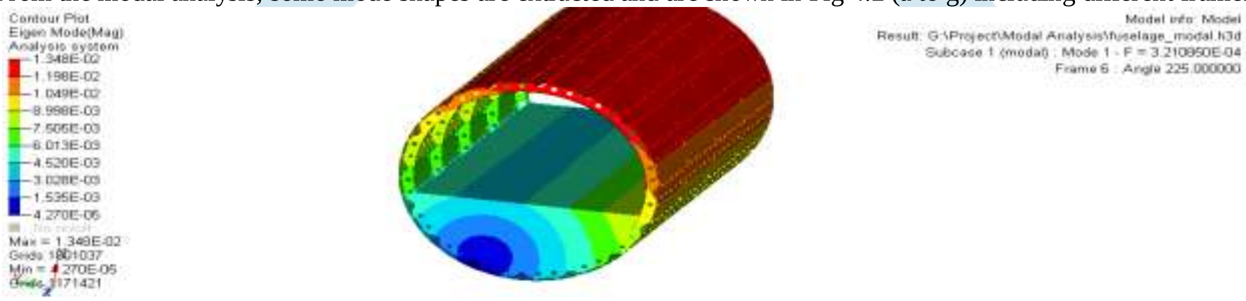

Fig 4.2(a) Mode shape 1 frame 6

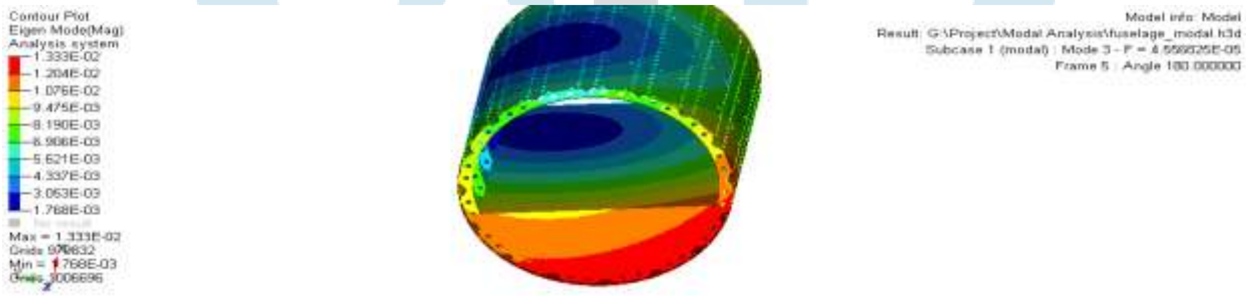

Fig 4.2(b) Mode shape 3frame 5

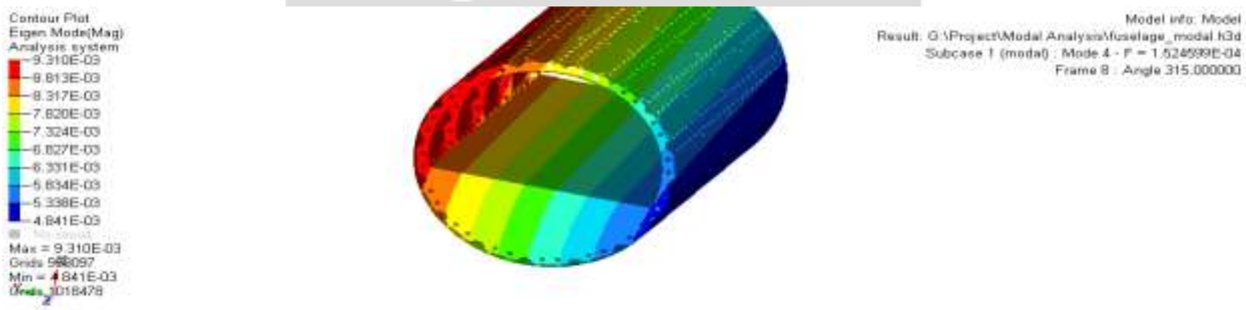

Fig 4.2(c) Mode shape 4 frame 8

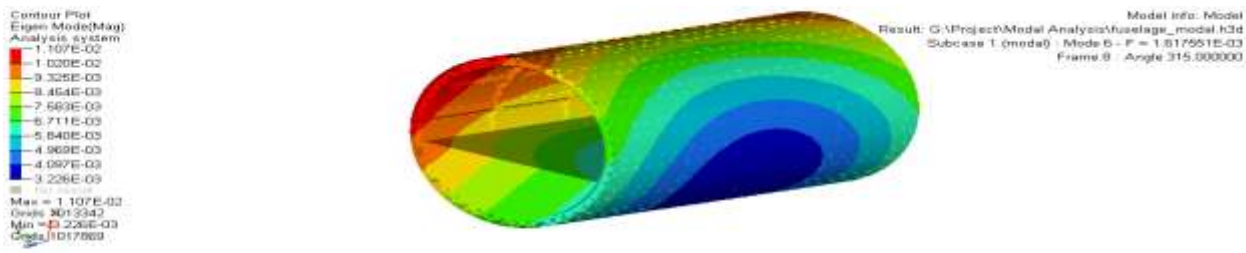

Fig 4.2(d) Mode shape 6 frame 8

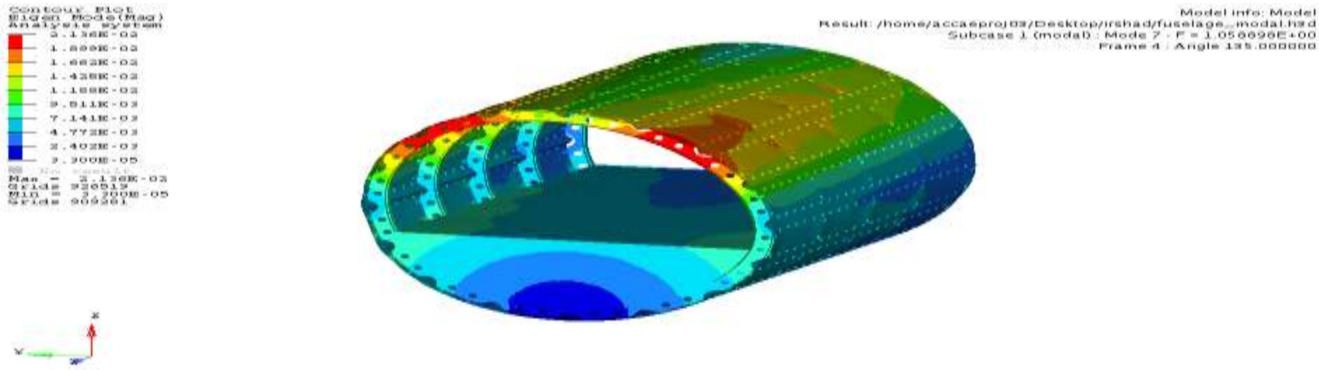

Fig 4.2(e) Mode shape7 (Bending) frame 4

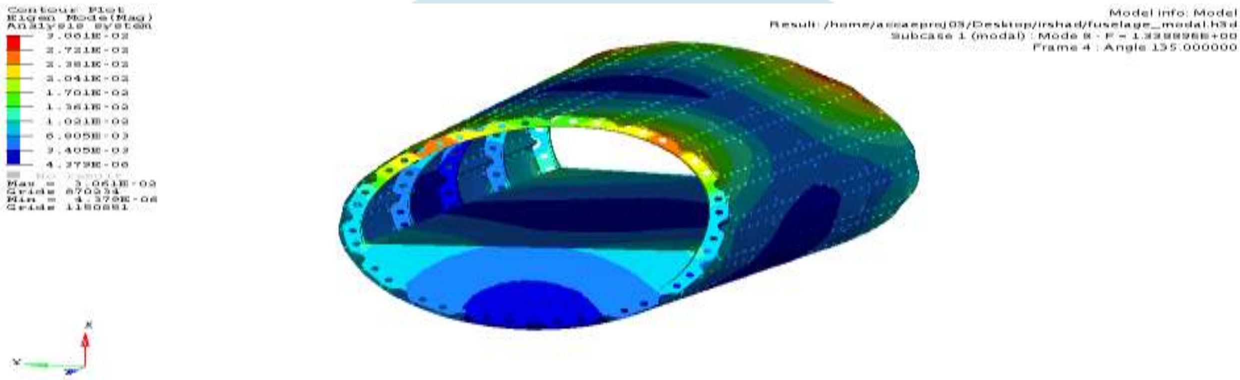

Fig 4.2(f) Mode shape 8 (Twisting)frame 4

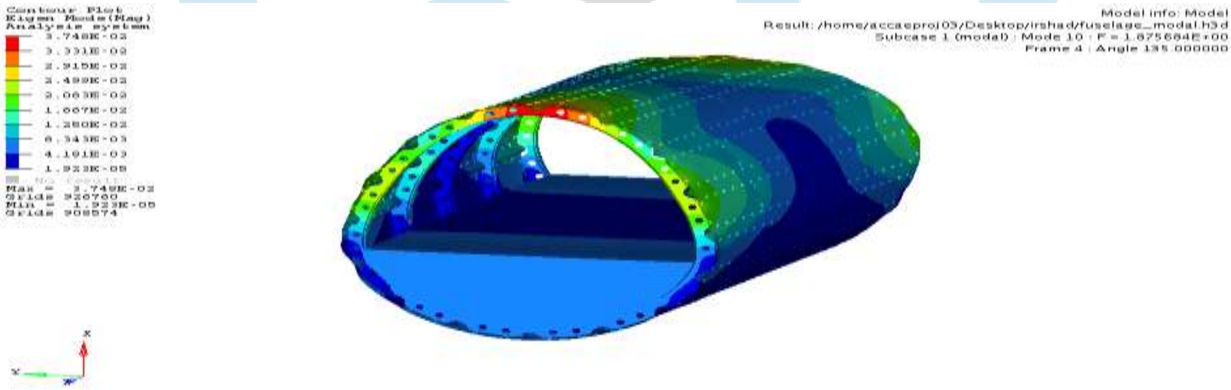

Fig 4.2(g) Mode 10 Frame 4(Bending +Twisting)

Mode shapes had found to be rigid body values, andrepresent the individual bending and twisting of the assembly model and bending twisting both on final mode .

Natural frequencies of the mode shapes are obtained and it is noted that from mode shape 1 to 8 it has varying values but after mode shape 8 it is almost the same. These values are tabulated in Table 4.1.

Table 4.1: Representation of Values for Modal Analysis

Sub case	Mode	Frequency	Eigen Value	Stiffness	Mass
1	1	3.210E-04 Hz	1.348E-02	1.348E-02	1.0
1	2	2.693E-04 Hz	1.328E-02	1.328E-02	1.0
1	3	4.556E-05 Hz	1.334E-02	1.334E-02	1.0
1	4	1.524E-04 Hz	9.310E-02	9.310E-02	1.0
1	5	7.613E-04 Hz	1.149E-03	1.149E-03	1.0
1	6	1.617E-03 Hz	1.107E-02	1.107E-02	1.0
1	7	1.058E+00 Hz	2.136E-02	2.136E-02	1.0
1	8	1.338E+00 Hz	3.061E-02	3.061E-02	1.0
1	9	1.823E+00 Hz	2.569E-02	2.569E-02	1.0
1	10	1.875E+00 Hz	3.748E-02	3.748E-02	1.0

4.3 Converged Graph Convergence of the model took eight iterations for solving the optimization starting from 1000 MPa to 227 MPa it has converged. The converged graph is shown in Fig 4.3.

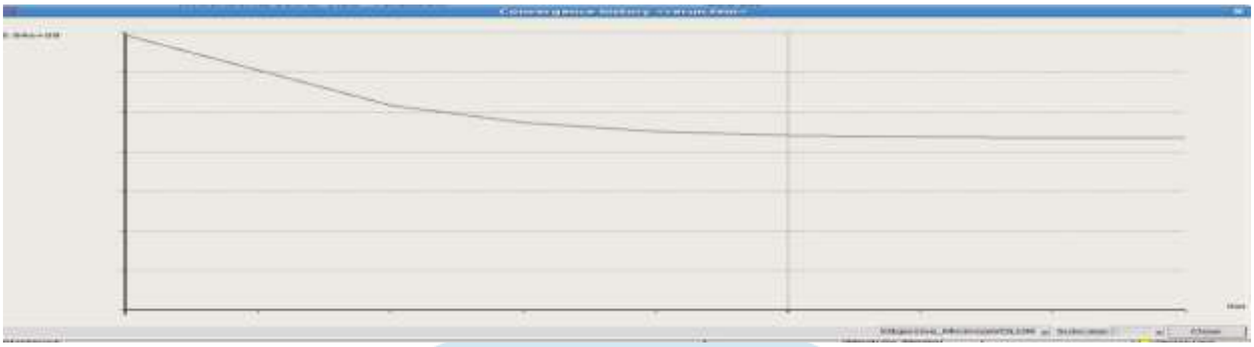

Fig 4.3 Topology optimization converging graph

Fig 4.3.1 and 4.3.2 shows the final optimized model. Therefore the design of the platform can be modeled which also carries the safe loads under applied load.

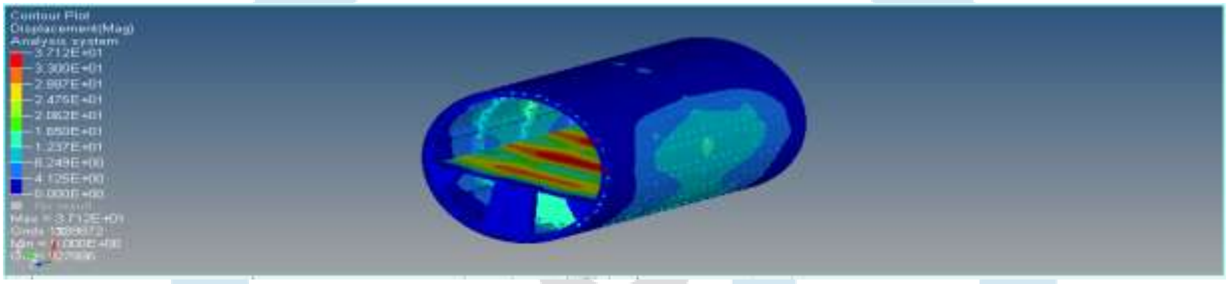

Fig 4.3.1: Optimized model

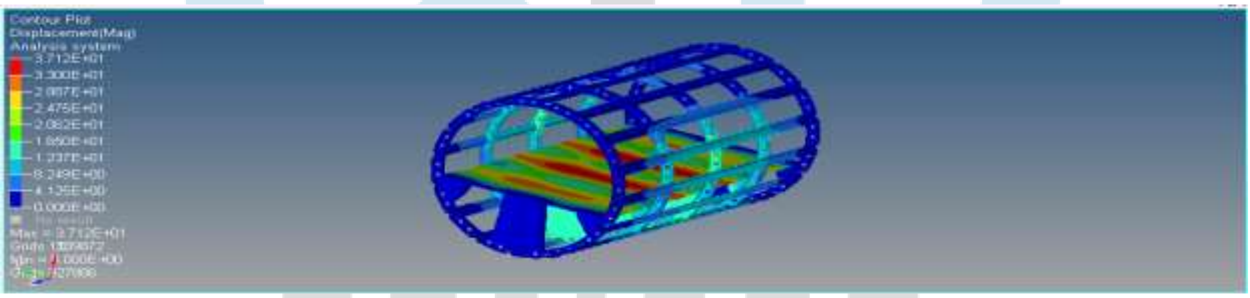

Fig 4.3.2 Optimized model close-up view

V. CONCLUSION

Fuselage assembly including skin, frame, stringers, rivets and rivet frame was modeled in Catia V5.From structural analysis maximum stress (227.4MPa) was found at platform and minimum stress is found at frame.Form the modal analysis, mode shapes from 1 to 6 represents rigid body values and 7 to 10 represents the bending twisting and both. The topology optimization, the new- shaped cross section is showing most deserved results when compared to T- shaped cross section.

Further dynamic analysis can be performed to account the twisting and bending.Transient analysis can be performed.

REFERENCES

[1] B. Y. Kolesnikov, L. Herbeck, "Carbon Fiber Composite Airplane Fuselage: Concept and Analysis", Section II, Possible Ways of Russia-Europe Cooperation. April 2001, page 1-11.
[2] Rozvany, G. I. N., A critical review of established methods of structural topology optimization, Structural and Multidisciplinary Optimization, 37, 217-237, 2001.
[3] Tomlin, Matthew, and Jonathan Meyer. "Topology optimization of an additive layer manufactured (ALM) aerospace part." In Proceeding of the 7th Altair CAE technology conference, pp. 1-9. 2011.
[4] W. Rust, M. Kracht, J. Overberg, Experiences with ANSYS in Ultimate-Load Analyses of Aircraft Fuselage Panels, Proceedings of the 2006 International ANSYS Conference, Pittsburgh, May 2006.
[5] Zolekar, Vivek, and Dr LN Wankhade. "Finite Element Analysis and Optimization of IC Engine Piston Using RADIOSS and OptiStruct." In Altair technology conference. 2013
[6] Kulkarni, Vinayak, Anil Jadhav, and P. Basker. "Finite Element Analysis and topology Optimization of Lower Arm of Double Wishbone Suspension using RADIOSS and Optistruct." International Journal of Science and Research (IJSR) 3, no. 5 (2014).
[7] Bendsoe, M. P., and Sigmund, O., "Material interpolation schemes in topology optimization," Archive of Applied Mechanics, 69 (9-10), 635-654, 1999.