Design Optimization of Rotor Craft Horizontal Tail Plane using Fea

P Muralikrishna
PG Student,
PVP Siddhartha Institute of Technology,
Vijayawada, India

U Koteswara Rao
Associate Professor,
PVP Siddhartha Institute of Technology,
Vijayawada, India

R. Vijayakumar
Rotary Wing R&D Centre,
Hindustan Aeronautics Limited,
Bangalore, India.

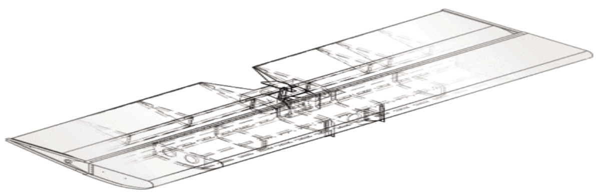
Abstract **- An accurate finite element model for optimum design of Horizontal Tail Plane (HTP) has been developed by predicting the strength and stiffness under critical flight loads. In the preliminary design phase computational studies are carried for HTP made of Aluminum and then the analysis is extended for composite HTP. Composites are a type of material that generally combines two materials yielding mechanical properties that are different than its constituent parts. These constituents are classified as either a fiber or a matrix. The main objective of this project is to design a carbon-fiber composite spar for weight optimization that meets the strength requirement. The spar is of an I-beam consists of carbon fiber unidirectional and woven laminae, as well as Nomex honeycomb to stiffen the structure. The detailed design and analysis of top and bottom composite sandwich panels and centre spar with all other ribs is carried out for sizing of the HTP. The spar geometry with optimized stacking sequence in the top and bottom flange and web is arrived by identifying the critical elements for strength improvements. The geometrical modeling is carried out by a modeling tool Unigraphics NX7.5 and FE modeling by the pre-processor of MSc Patran. The element type used for meshing was 2D shell elements with QUAD4 element topology and 1D element with beam topology. Static analysis done using MSc Nastran.**

Key words: Rotor craft, Horizontal tail plane (HTP), Sandwich panel, Spar, Ribs

1. INTRODUCTION:

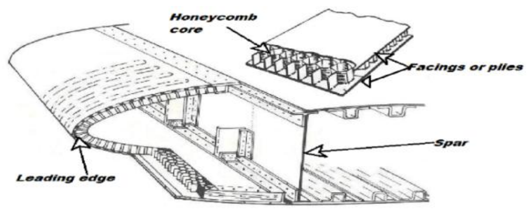
A typical Rotorcraft consists of several structural modules namely Fuselage, Main and Tail Rotor, Transmission system, Power plant, Landing gear and Tail boom with Empennage (Empennage Consists of Horizontal Tail Plane, Vertical Fin, Tail rotor and End Plate). Horizontal Tail Plane is a fixed stabilizing surface at a pre-determined angle of attack. It is a small lifting surface located on the tail (empennage) behind the main lifting surfaces of a fixed-wing aircraft as well as other non-fixed-wing aircraft such as helicopters and gyroplanes. Generally the HTP is attached to two frames of the tail-boom by bolts and elastomeric damper for controlling vibrations. It gives pitch stability to the helicopter in forward flight. The tail plane provides stability and control. In addition the tail-plane helps adjust for changes in the centre of pressure, and centre of gravity caused by changes in speed and attitude, or when fuel is burned off, or when cargo or payload is dropped from rotorcraft. Geometrically it has an inverted aerofoil cross section throughout the span. It consists of a centre spar which runs from tip to tip. The top and bottom shells having sandwich construction which is made out of metallic/composite face sheets and nomex (ox) core. It has several ribs across the span.

The primary factors to consider in helicopter structures are strength, weight, and reliability. These factors determine the requirements to be met by any material used to construct or repair the helicopter. Airframes must be strong and light in weight. The helicopter built so heavy that it couldn't support more than a few hundred kilograms of additional weight would be useless. All materials used to construct an aircraft must be reliable. Reliability minimizes the possibility of dangerous and unexpected failures. The constructional (material location) and part details of the HTP was clearly shown in the Figure.1. It shows the all components like top shell, bottom shell, center spar and ribs. The figure below will also describes the cross-sectional details and their material stacking up sequence.



a) Schematic representation of HTP

b)



c) Constructional details of HTP

Fig.1 Geometry of Horizontal Tail Plane

The helicopter must possess positive static longitudinal control force stability at critical combinations of speed stability (static longitudinal stability). The objective is a moment that will be nose-up (positive) with increasing and nose-down (negative) with increasing speed. The parasitic drag of the fuselage and landing gear provides negative speed stability. The horizontal tail surfaces produce an increasing down force with increase in speed and this tends to maintain the fuselage attitude relatively constant over a large part of the speed range. After a speed increase therefore the cyclic stick will be further forward but the nose attitude will be little different once the aircraft has accelerated.

Polagangu James, D. Murali Krishna said [1] Inter spar ribs of wing of a transport aircraft is subjected to various types of loads. One of the loads that pose stability problem to the inters par ribs of a wing is brazier load, which arises due to flexure of the wing. This paper describes about the finite element analysis of inter spar ribs of a wing at local level against brazier load. This study has been taken place while converting metal wing in to composite wing. The objective of this study is to reduce the weight penalty to the maximum possible extent by removing material wherever feasible. This paper is limited to discuss about the linear buckling analysis of ribs against brazier load. The buckling factor of ribs under consideration are reported in terms of square root times the eigenvalue obtained from finite element analysis, which represent the nonlinear effect of bending moment on brazier load. This study has helped to reconfigure/redesign the inter spar ribs of wing. This has led to substantial weight saving of 2.85 Kg which accounts 15.77% reductions of total mass of inter spar ribs.

As per Vinod S. Muchchandi, S. C. Pilli [2] vertical tail and the rudder are important structural components of an aircraft. Movement of the rudder controls the yawing of an aircraft. Structurally speaking vertical tail is a typical mini-wing construction. A major difference could be absence of ribs and multiple spars (more than 2) in the vertical tail construction. Vertical tails have. Symmetrical airfoil cross sections. Therefore in the absence of rudder deflection there is no aerodynamic load acting on the fuselage. However significant side loads develop due to rudder deflection and this is the major design load for the vertical tail. For transport aircraft side gust load is also important from a design point of view. In this project a typical spar of a vertical tail of a transport aircraft will be analyzed. Loads representative of a small transport aircraft will be considered in this study. An efficient tapered spar beam will be designed for this load. SOM approach will be used for preliminary sizing of the spar. This will be followed by FEA for a more accurate stress analysis.

Prabha C, Nandakumar C.G [3] the concept behind sandwich construction and its application in naval and commercial fields is highlighted with special reference to underwater shell forms. Also examines the various shell finite elements generated by researchers for the analysis of sandwich shells. A comprehensive overview of finite elements available for analysis of sandwich structures in commercial software packages is presented. The accuracy of the axisymmetric finite element sandwich shell is evaluated by comparing the results with the numerical results available by analysing the shell using ANSYS. The results of reasonable accuracy have been realized at the cost of heavy computation using the software. The need for a sandwich shell finite element based on sandwich shell theory has been justified.

Chandrakala, A.B, Shivalingappa, D. Halesh Koti [4] to present the preliminary design process of a spar beam of an aircraft wing structure. Wings of the aircraft are normally attached to the fuselage at the root of the wing. This makes the wing spar beam to behave almost like a cantilever beam. A six seated transport aircraft wing spar design is considered and the initial design has been carried out using conventional design approach which was found out to be with heavy weight. In the next stage, as the primary load carrying ability is required in bending, the design was carried out as per the external bending moment at each station. A finite element approach was used to calculate the stresses developed at each station for a given bending moment. Several iterations were carried out for design optimization of the spar beam. A typical aluminum 2024-T3 was chosen as the material for spar beam and linear static analysis was used for the stress analysis. The spar beam was designed to yield at the design limit load and material saving was achieved through the design optimization.

Shabeer KP1 , Murtaza M A [5] to develop an accurate model for optimal design through design the structure of wing that combine the composite (Skins) and isotropic materials (all other structures) and compare this with the same wing made by changing the orientation of composite ply orientation in skin. The optimum design for each wing with different ply orientation can be obtained by comparing stress and displacement. Structural modeling is completed with the help of CATIA V5; each components modeled separately and assembled using Assembly workbench of CATIAV5, this assembly is then converted to IGS file. Finite element modeling is completed in MSc Patran using the IGS file as geometry, the element type used for meshing was 2D shell elements with QUAD4 element topology and different parts are connected using RBE2 connection. Static analysis done using MSc Nastran. The finite element model obtained is analyzed by applying an inertia force of 1g and then aerodynamic result (lift) is used to simulate the wing loading on the wings. Optimum design is found by tabulating stress and displacement for each ply combination.

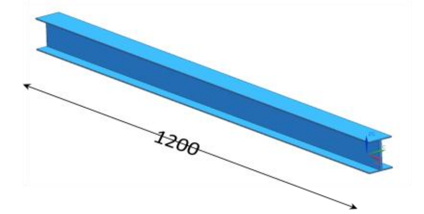
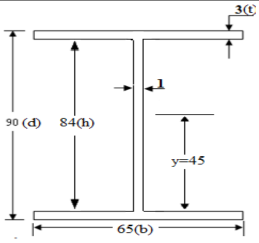
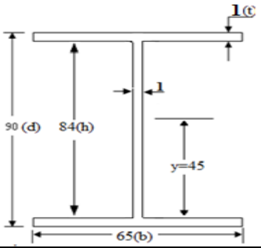
2.0 DESIGN AND ANALYSIS OF HTP-VALIDATION STUDIES

Prior to the design and analysis of HTP few validation studies are carried out for spar and panel.

2.1 Spar

The geometry of the spar with two different cross sections are considered. (Ref.Table1)

Table- 1 Dimensional details of spar for two cases

	Geometry
Spar	
Section 1	
Section 2	

All dimensions are in mm

The material properties and load details are given in table-2.

Table-2 Material properties and loading

Load on the spar	P=3000N
Young's modulus	E = 70000 N/mm^2
Density	ρ = 2700 kg/m^3

By assuming the total width 600mm of the aerofoil section for a length of the spar 1200 mm the corresponding pressure p is given by

$$p' = \frac{force}{surface\ area}$$

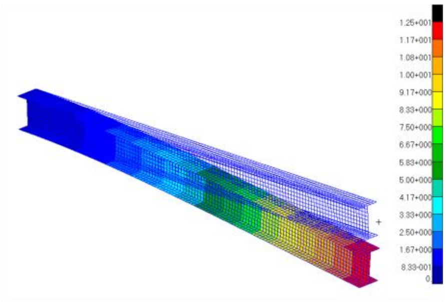
$$= \frac{3000}{600x1200} = 4.166x10^{-3}\ \text{MPa}$$

Assuming the spar as cantilever beam subjected to transformed Uniformly Distributed Load (UDL), then pressure p" is given by

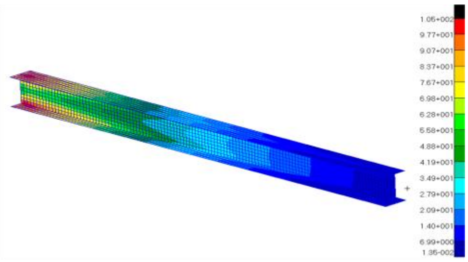
p"= p' * w

$$= 4.16x10^{-3}x600 = 2.5\ \text{N/mm}$$

Case 1: Spar with section 1 (ref. Table 1)



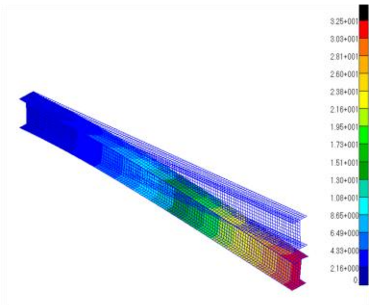
Displacement Plot



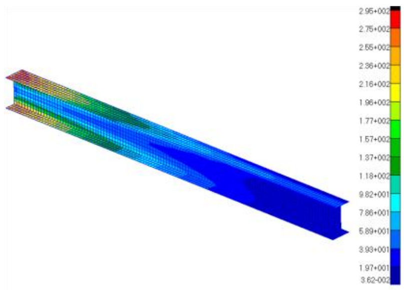
Vonmises Stress Plot

Fig.2 Contour plot of spar with section 1

Case 2: Spar with section 2 (ref. Table 1)



Displacement Plot



Vonmises Stress Plot

Fig.3 Contour plot of spar with section 2

When applying the 2.5 uniformly distributed load on spar by fixing the one end gives the following results for two cases.

Analytical and FEM are summerised and compared in Table.4

Table-3 Results comparision

	VonMises stress, MPa		Displacement, mm		Weight, kg
	FEM	Analytical	FEM	Analytical	
Case 1	106	102	12.5	11.75	1.53
Case 2	296.3	257.7	32.5	29.4	0.705

2.2 Plate under Distributed Transverse Load

Consider a thin rectangular plate of dimensions a x b, simply supported along each of its four edges and carrying a distributed load q(x, y)

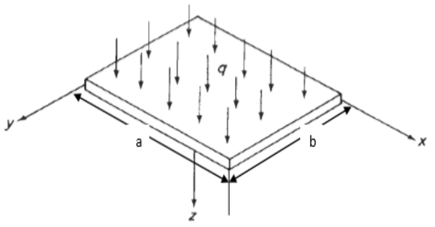


Fig 4.plate under uniform pressure

$$(W_{x=0}) = 0 \quad \text{and} \quad \left(\frac{\partial^2 w}{\partial x^2}\right) = 0 \qquad 2.6$$

The free edge:

Along a free edge there are no bending moments, twisting moments or vertical shearing forces, so that if x = 0 is the free edge then

$(Mx)_{x=0} = 0$ $(Mxy)_{x=0} = 0$ $(Qx)_{x=0} = 0$

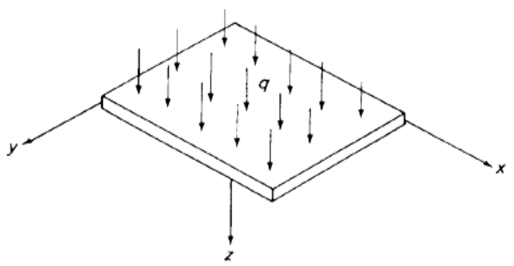


Fig.5 Plate under normal pressure

$$Wmax = \frac{16q}{\pi^6 D} \sum_{m=1,3,5}^{\infty} \sum_{n=1,3,5}^{\infty} \frac{\sin(m\pi x/a)\sin(n\pi y/b)}{mn\left[\frac{m^2}{a^2} + n^2/b^2\right]}$$

$$W_{max} = 0.0443 q_0 \frac{a^4}{Et^3}$$

FE model of plate is shown in figure 5

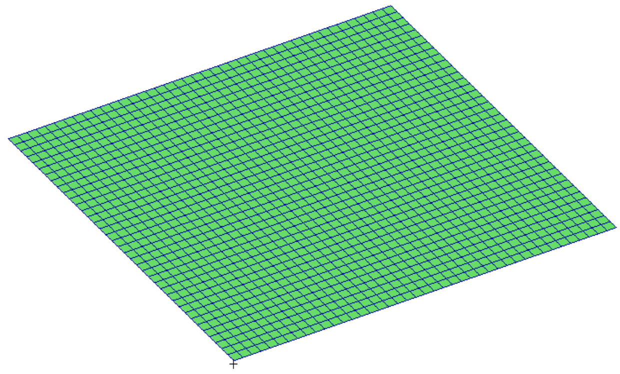


Fig.6 FE model of plate is shown in

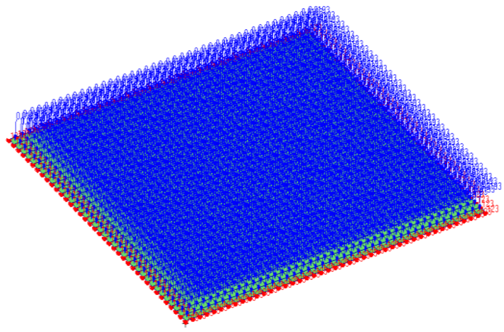
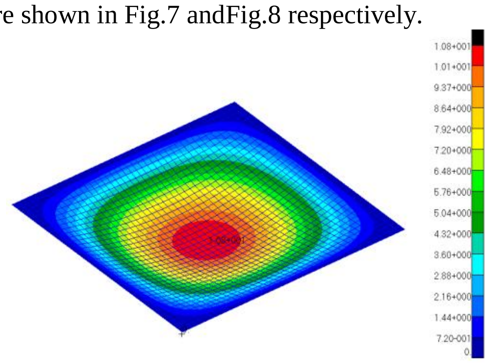
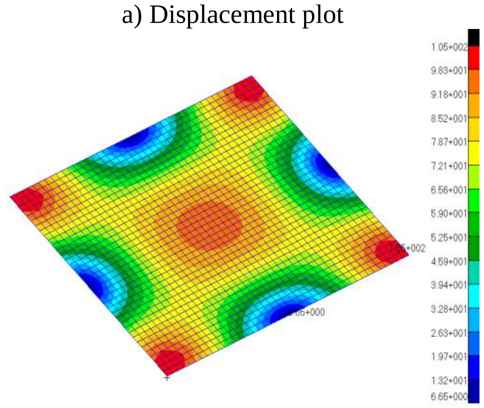


Fig.7 FE model with B/C

The displacement and stress plots for 3mm and 2 mm thick plate are shown in Fig.7 andFig.8 respectively.

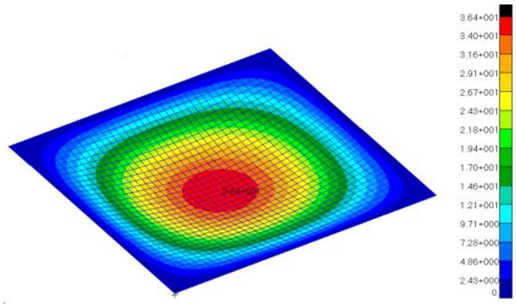


a) Displacement plot

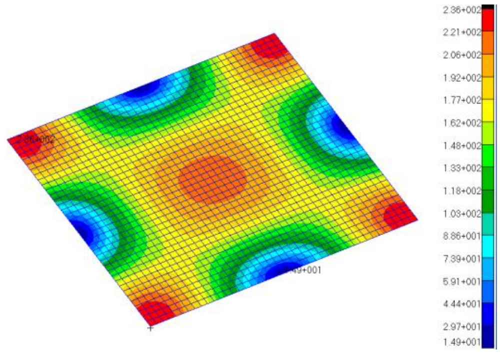


b) von Mises plot

Fig.8 displacement and vonmises plots of 2mm thick plate



a) Displacement plot



b) von Mises plot

Fig.9 displacement and vonmises plots for 3mm thick plate

The results summery from analytical and FEM are given in Table 4

Table-4 Conclusion of Results

Plate thickness	Von Mises stress In MPa		Displacement in mm		Weight in kg
	FEM	Analytical	FEM	Analytical	
3 mm	106	106.31	10.8	11.02	1.259
2 mm	236	239.9	36.4	37.21	0.839

3.0 FINITE ELEMENT MODELLING FOR STRESS ANALYSIS OF HTP

The FE modeling of HTP is carried out by using pre-processing capabilities of commercially available FE code MSc PATRAN. Top and bottom panels of HTP have been idealized by with QUAD 4 shell elements. Spar webs were modeled by using 2D shell element in order to get torsion response where the spar and rib flanges were modeled using 1D beam elements to get more accurate results in bending responses. Rib webs have been idealized with 2D shear elements. Typical FE model is shown in Fig.9. To minimize the computational cost model symmetry is considered.

The symmetry boundary conditions have been provided at the leading root edge of the root rib by fixing three translations and rotational degrees of freedom as shown in Fig.10

The structural modeling of the designed wing will be conducted by using the Finite Element Method and MSC®/PATRAN Package program. This chapter describes the selection of the element types, solid modeling for FEM, part connection methods, mesh generation and the boundary conditions.
The most commonly used element type in aerospace industry is the two-dimensional (2D) [9] elements since the industry is mostly dealing with thin walled structures. Almost every main structural part can be modeled using this type of elements. 1D type elements on the other hand also have applications for connecting apparatus and mostly for beamlike structures. 2D type elements are the most commonly used element type in aviation industry. This type of elements generally designed for thin walled structures such as the skin, however, for the wing considered the thicknesses of the other members of the torque box were also small. Thus, those members could also be considered as thin walled structures and were modeled by using 2D type elements. When the spars were taken into account, it was determined that the webs of the spars should be modeled by using 2D shell element in order to get torsion response where the flanges should be modeled using 1D beam elements to get more accurate results in bending responses.

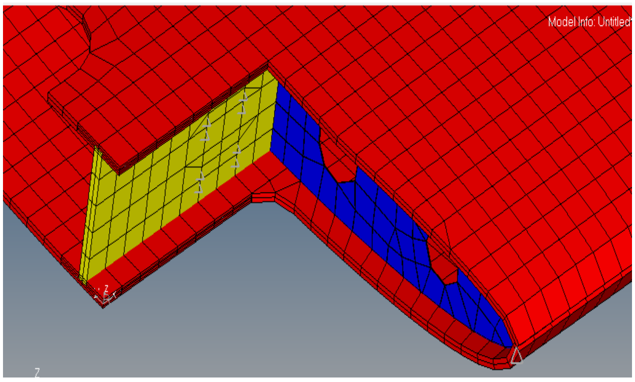


Fig.10 Typical FE mesh of HTP

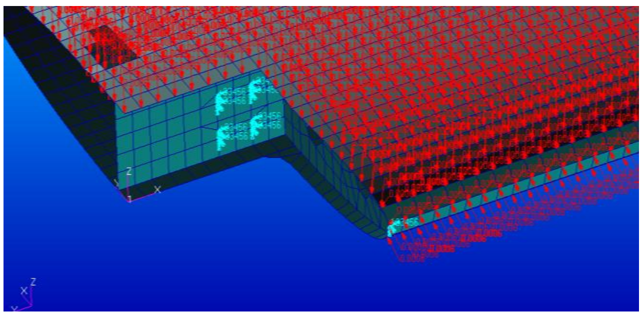


Fig.11 Boundary Conditions

The geometrical modeling of the main structural members were done by generating surfaces at the mid-planes of the structural parts for which 2D shell elements were assigned. Then, the edges of the surfaces generated for the spar webs were used to create the 1D beam elements assigned for the spar flanges. First the geometrical model of the composite skin was generated. Figure gives the isometric view of the surfaces representing the –skin of the wing torque box designed.

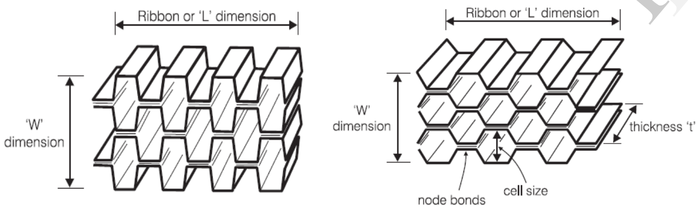
Table-5 Summary of selection of element type

Spar web	2D Shell
Spar flange	1D Beam
Skin	2D Shell
Rib	2D Shell

The connection between spar, ribs and spar flanges were connected by taking the 1D element of T-cross section for getting the good results about bending of spar flange elements and mentioning the element connection between them.

The boundary conditions applied for metallic and composite HTP were the same. The root end of HTP was arrested for 3 translations and 3 rotational degrees of freedom. The load applied on the component was a pressure load which is in terms of a magnitude by taking the ratio of force to applied surface area.

It can withstand and absorb more stress or energy in the direction of thickness only, that's why it has more strength modulus in the direction of thickness. The properties are shown in the below.



a) OX core b) Nomex core

Fig. 12 Schematic representation of core

The [6] core plays a vital role in the analysis of this HTP, Nomex core is used for the spar section and OX core is used for top shell (Fig.10) of the composite panel because it is a stretched nomex armid core which is used only on curved geometrical parts and aerodynamically shaped objects to get to fit on the surface.

The material properties those used for skin and spar sections should be high strength to weight ration. The [7] Kevlar and carbon epoxy woven fabric materials have their unique properties to bear the high strength. Structures that combine high stiffness, strength, and mechanical energy absorption with low weight are needed for aerospace applications. Sandwich structures are the most impressive and viable option because of its weight saving potential. Sandwich construction can be described as hybrid construction consisting of two outer dense facings with a light weight core. The facings usually provide the load carrying capability for sandwich construction. A sandwich construction is characterized by very high flexural strength, low lateral deformations, higher buckling resistance and higher natural frequencies. The high stiffness to weight ratio makes this kind of structural element a very attractive design option in weight critical structures. The thickness of a structure can be increased by introducing a low modulus core in between the laminates even without incurring the weight penalty from adding extra laminate layers or stiffeners. In effect the light weight core acts as a separator between the load-bearing facings also.

3.1 Loading Conditions for HTP

The root end of HTP was fixed for 3 translational and 3 rotational degrees of freedom by selecting four nodes of spar near to the root end in the place of bolted joint provided to assemble the horizontal tail plane to the empennage of helicopter as in realistic.

The load applied on HTP is taken as 3125 N by considering the helicopter velocity 240kmph and density of air is 1.3 kg/m^3 at 0^{0}c of temperature. [10]

Table-6 Material properties of Kevlar/epoxy and carbon/epoxy

Property	Carbon	Kevlar	Units
Ply thickness	0.125	0.1	mm
Density	1.58E-06	1.38E-06	kg/mm^3
Ply longitudinal modulus	142	75.8	GPa
Ply transverse modulus	10.3	8.8	GPa
Ply Poisson's ratio	0.27	0.34	
Ply shear modulus in plane	7.12	2.07	GPa
Ply transverse modulus parallel to fiber direction	3.15	1.29	GPa
Ply transverse modulus perpendicular to fiber direction	7.12	2.07	GPa
Ply longitudinal tensile strength	1.83	1.38	GPa
Ply longitudinal compressive strength	1.09	0.58	GPa
Ply transverse tensile strength	0.05	0.03	GPa
Ply transverse compression strength	0.22	0.13	GPa
Ply shear strength	0.07	0.04	GPa

Table-7 core material properties

Product construction		Compression		Plate shear			
ρ	t	Stabilized		L direction		W direction	
Kg/m^3	mm	σ (MPa)	G (MPa)	σ (MPa)	G (MPa)	σ (MPa)	G (MPa)
No-mex 64	6	5.0	190	1.55	55	0.86	33
OX 48	5	2.9	120	0.8	20	0.85	35

Where

ρ - Density
σ – Strength
t – Cell size

4.0 DISCUSSION OF RESULTS

The HTP of a typical helicopter empennage is analyzed for metallic as well as composite construction separately with finite element program MSC Patran as well. Higher mesh density with good quality of elements is used for finite element modeling. Loads and boundary conditions are accurately simulated to obtain the realistic loading conditions.

The stress and displacement plots of metallic HTP is shown in Fig. 12 and Fig.13 Based on the results of finite element predictions and by the calculations of the stress analysis approach, it is apparent that the material isotropy has more effect on the top and bottom flange. From the anaysis the maximum stress is compared with yield stress and ultimate stress of 2024 T351

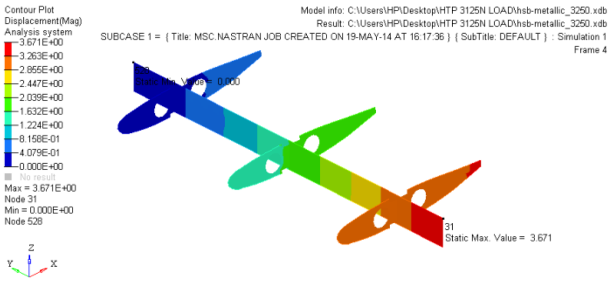


Fig.13 Displacement of metallic spar

aluminum alloy. The obtained magnitudes of maximum stress are less than the yield stress and ultimate stress so we conclude that material in elastic limit and not yet started yielding. The stresses induced in metallic HTP structural members i.e., spar, top and bottom shells and ribs are within the allowable stress range and margin of safety if 2.3 for the metallic HTP. The total weight of the entire HTP structure is 7.69 kg.

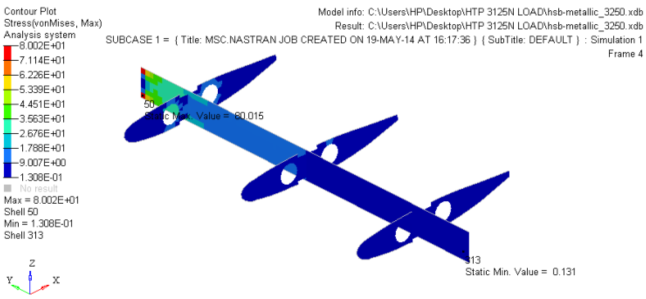


Fig.14 Vonmises stress of metallic spar

As part of weight reduction an attempt is made by an analysis for satisfying the aerodynamic performance and by converting the metallic structure to composite structure in the combination of monolith and sandwich parts. Here, optimization studies are carried out rigorously to meet the best optimum configuration. In the analysis, material properties [6] used is the ones obtained from mechanical testing. The stress contour plot of composite HTP using carbon and Kevlar layers and displacement plot are shown in Figures 13 and 14. Maximum stress is observed at bottom flange segment of spar near root end.

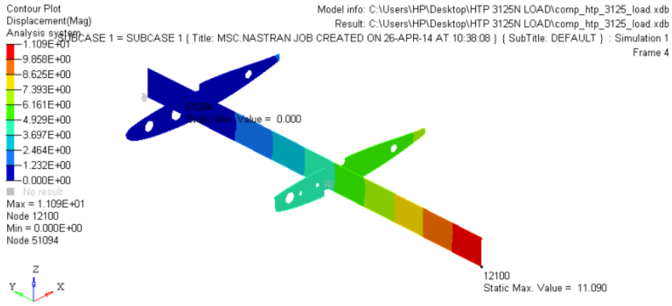


Fig.15 Displacement of composite spar

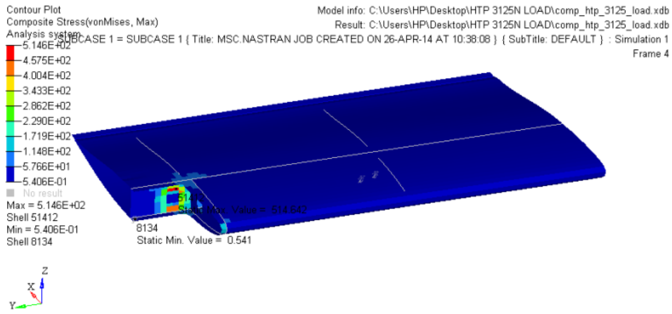


Fig.16 VonMises stress of composite HTP

In the present analysis HTP sandwich panels are analyzed for face sheet wrinkling and checked for fiber and matrix strength for the critical forces obtained from FE analysis. Spar and ribs are verified for web shear buckling apart from fiber and matrix strengths. The Spar is major load carrying member which takes load from skin.

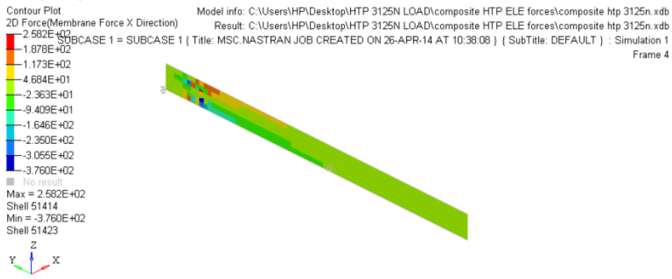


Fig.17 Max 2D element force at 51423 element

The spar is attached to Helicopter Tail boom using high stiff brackets and bolts. The margin of safety for spar section is 0.27 at element number 51423, so the structure will withstand for the applied load. Element forces were, where the element get lowest margin of safety and those are used for finding the reserve factor by using MATHCAD software. The first most layer of 45^0 was had 1.41 reserve factor for matrix, so it can withstand the applied load.

It can be observed from the results that, in terms of deflection and stress the results are varied due to material orthotropic. Thus, the stiffness of the spar is improved by converting to composite construction. It should also be stated that, the results of composite HTP analysis are not close to the stresses and deflections found in the metallic HTP analysis. The reason for this is the improved stiffness due to orthotropic material construction for same load is applied in both cases. Here, one can expect to have variation of results in terms of the stresses as well. Overall, the optimized solution for the stated loading conditions on composite HTP gives the total weight 4.32 kgs.

5.0 CONCLUSIONS

In this paper a HTP structure of a typical helicopter, subjected to critical flight loads was studied. The HTP was structurally modeled by using MSC®/PATRAN package program and the structural analyses were conducted by the help of the MSC®/NASTRAN package program. During the analysis the required design exercises were carried out by using the Computer Aided Design tool Unigraphics NX7.5. The optimized HTP was then analyzed to verify the strength requirements. Furthermore, the results obtained from the design experiments were used for the verification and the tuning of the initially developed structural model. Finally, the verified structural model was again structurally analyzed to obtain the final results. The maximum element force was observed at bottom flange of spar root end. The total weight is reduced about 43.8 %. The spar gets 1.93 of margin of safety so it can withstand the maximum load which is applied on it.

The reserve factor for top most layer of 45 degree was 2.21.Based on the results of finite element predictions and by the calculations of the stress analysis approach, it is apparent that the air drag load has more effect on the top and bottom flange. From the load cases the maximum stress is compared with yield stress and ultimate stress of 2024 T351 aluminum alloy. The obtained magnitudes of maximum stress are less than the yield stress and ultimate stress so we conclude that material in elastic limit and not yet started yielding.

REFERENCES

1. Polagangu James*, D. Murali Krishna, Gaddikeri Kotresh and Byji Varughese Advanced Composites Division, National Aerospace Laboratories, Bangalore-560017, INDIA (Finite Element Analysis of Inter Spar Ribs of Composite Wing of Light Transport Aircraft against Brazier Load)
2. Vinod S. Muchchandi1, S. C. Pilli2 P.G. Student, Department of Mechanical Engineering, KLE Dr.MSS CET, Belgaum, Karnataka, India1 Principal, Department of Mechanical Engineering, KLE Dr.MSS CET, Belgaum, Karnataka, India2 (Design and Analysis of A Spar Beam For The Vertical Tail of A Transport Aircraft).
3. Piranha C, Nandakumar C.G. Department of Ship Technology, Cochin University of Science and Technology, Kerala, India. (Structural Analysis of Sandwich Submarine Structures)
4. Chandrakala, A.B, Shivalingappa, D. Halesh Koti 1Department of Mechanical Engineering, Adhiyamaan College of Engineering, Hosur, India (preliminary design of an aircraft wing spar)
5. Shabeer KP1 , Murtaza M A2 PG student, Department of Mechanical Engineering, The Oxford College of Engineering, Bangalore, Karnataka, India1 Professor, Department Of Mechanical Engineering, the Oxford College Of Engineering, Bangalore, Karnataka, India2 (optimization of aircraft wing with composite material)
6. Hexcel technical data sheet from Google (http://www.matweb.com/search/GetMatlsByManufacturer.aspx?manID=81)
7. Aswin Sheshadri, Design and analysis of composite beam for side impact protection of occupants in a sedan,
8. Theory of plates by S.timoshenko, Professor Emeritus of engineering mechanics, Stanford University.
9. Levent Unlusony, structural design and analysis of machine adaptive wing for unmanned air vehicle, the graduate school of natural and applied science.
10. Stephen Haley, Design Optimization and Verification of a Horizontal Stabilizer for the SeaStryder600 Wing-In-Ground-Effect (WIG) Aircraft, Graduate Department of Mechanical and Industrial Engineering University of Toronto

.