

January 2012

Design and Analysis of Propellant Tanks Support Structure for an Advanced Spacecraft

Sk. J. Basha

Structures Group, ISRO Satellite centre, Bengaluru, India, basha@isac.gov.in

Follow this and additional works at: <https://www.interscience.in/ijarme>



Part of the [Aerospace Engineering Commons](#), and the [Mechanical Engineering Commons](#)

Recommended Citation

Basha, Sk. J. (2012) "Design and Analysis of Propellant Tanks Support Structure for an Advanced Spacecraft," *International Journal of Applied Research in Mechanical Engineering*: Vol. 1 : Iss. 3 , Article 9.

DOI: 10.47893/IJARME.2012.1035

Available at: <https://www.interscience.in/ijarme/vol1/iss3/9>

This Article is brought to you for free and open access by the Interscience Journals at Interscience Research Network. It has been accepted for inclusion in International Journal of Applied Research in Mechanical Engineering by an authorized editor of Interscience Research Network. For more information, please contact sritampatnaik@gmail.com.

Design and Analysis of Propellant Tanks Support Structure for an Advanced Spacecraft

Abdul Khaleelullah¹, Sk. J. Basha² & H. K. Rangavittal³

^{1&3}BMS College of Engineering, Bengaluru, India

² Structures Group, ISRO Satellite centre, Bengaluru, India

E-mail : basha@isac.gov.in

Abstract - This paper proposes a new type of configuration to support the propellant tanks in large size spacecrafts. Research in space industry over the last decade is focused on reducing the launch cost of the spacecraft. The main objective of this project work is to develop new concept of propellant tanks support structure for 6 ton class spacecraft. This configuration has four propellant tanks (2 oxygen tanks & 2 fuel tanks) which are mounted in polar form outside the central thrust cylinder and are supported by a flat rigid panel and two struts at fixed end and flat deck at free end. The stiffness requirements of advanced spacecraft are $> 8\text{Hz}$, $>27\text{Hz}$ and $>30\text{Hz}$ for global lateral mode of the spacecraft, oxygen tanks axial mode and fuel tanks axial mode respectively. For support structure various configurations, were worked out to meet stiffness requirements. The configurations considered are Honeycomb Sandwich panel, Honeycomb Sandwich panel embedded with doublers at tank attachment zone, Honeycomb sandwich panel embedded with rectangle beam at tank attachment zone and Honeycomb sandwich panel with circular struts at tank attachment zone. Design iterations were carried out for above four configurations and suitable configuration was finalized. Static analysis was also carried out for the given load cases to determine strain, Hoffmann failure index, SR and MOS for the composite central thrust cylinder. The geometric modeling of the propellant tanks support structure and main parts of the advanced spacecraft for all the four configurations were modeled by using UNIGRAPHICS-Nx6® and analysis carried out using MSC/PATRAN®, MSC/NASTRAN®.

Keywords - Spacecraft structure, Propellant tanks, CFRP, Spacecraft configuration.

ABBREVIATIONS

SR - Strength ratio
BD - Bidirectional
MOS - Margins of safety
FEA - Finite Element Analysis
RBE2 - Rigid body element type 2
CFRP - Carbon fiber reinforced plastic
LC - Load case

NOMENCLATURE

$>$ - Greater than
 $<$ - Less than
 g - Acceleration due to gravity
 θ - Struts Angle

1. INTRODUCTION

Typically in 6 ton class spacecraft, the propellant mass will be ranging from 3 to 3.5 tons, based on the life of the spacecraft. In a conventional spacecraft, propellant tanks are situated one over the other inside the central thrust cylinder.

Tank diameter is equal to internal diameter of the central thrust cylinder and is supported around its periphery by flexible tabs. Hence in conventional configuration, the cylinder diameter is decided by tanks diameter, as a result the overall size of cuboid will be larger. The main objective of this work is to reduce cuboid size as well as center of gravity of spacecraft without compromising on payload accommodation volume and hence the structural mass. In view of this, a new concept of propellant tanks support structure is developed, in which diameter of the central thrust cylinder is reduced and total propellant mass is distributed to four tanks, the tanks are placed around the central thrust cylinder and are supported by a flat rigid panel.

As reduction in weight of Spacecraft structures can help in increase of payload capacity, reducing launching costs, materials used for Spacecraft application must be light in weight and retain their strength and stiffness throughout the satellite mission, hence materials like CFRP and honeycomb sandwich composite are considered. Dynamic loads during launch phase form major criteria for design. Hence the design goal is to meet the specified stiffness requirements.

The static and dynamic structural behavior of a support structure has been studied using Finite Element Analysis. Four design configurations were considered for the support structure and selected the optimum design. The support structure has been modeled using UNIGRAPHICS-NX6 software and FE analysis is carried out to compute the stiffness. Various design modifications have been done to improve stiffness and to meet the desired constraints. Static analysis was carried out to estimate the strength using Hoffmann failure criterion. The main design requirements are the natural frequency of the system should be greater than 8Hz, 27Hz and 30Hz for global lateral mode of spacecraft, oxygen and fuel tanks axial modes respectively; All the structural elements are idealized by 2D shell elements; The free vibration and static analysis is carried out using MSC/Patran and MSC/Nastran software.

In free vibration analysis the natural frequency and mode shapes of the support structure are obtained; Static analysis is carried out for the given load cases to determine the strain, Hoffmann failure index, SR and MOS in all zones of central thrust cylinder.

2. FEA AND CONFIGURATION STUDY

2.1 Geometric Modeling

An advanced spacecraft structure consists of two basic modules - main bus and payload module. The main bus supports the payloads by providing them the required orbit and altitude control, electric power, thermal control, mechanical support and two way command and data link to ground. The payload bus performs the mission goals of the satellite such as communication by radio links and earth imagery etc. The main bus assembly consists of CFRP Central thrust cylinder, propellant tanks, base panel, Vertical panels, top panel, interference ring, shear panels and fixed end support panel (Refer figure 2.1).

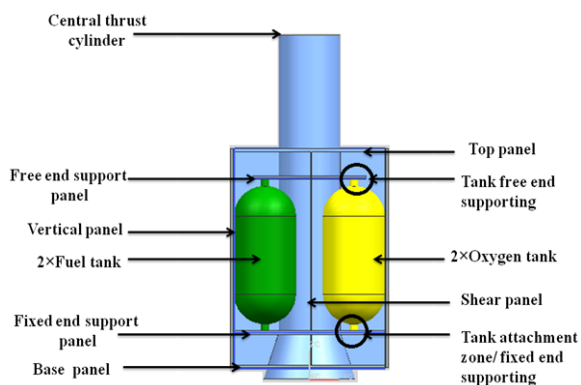


Fig. 2.1 Solid Model of Advanced Spacecraft

In this configuration, the propellant tank load bearing members are central thrust cylinder and fixed end support panel. Hence design and configuration study is carried out for these two structural components based on stiffness and strength. The configurations developed are as follows.

2.2 Finite Element Modeling

A finite element model is a complete idealization of the entire structural problem, including the node locations, element's physical and material properties, loads and boundary conditions. The model will be defined differently for analysis like normal mode analysis, linear static analysis and buckling analysis. The accuracy of the resulting solution will depend on how well the structure was modeled, the assumptions made for the load boundary conditions and accuracy of the elements used for a given problem. In general, the solution will be more accurate; as the model is subdivided into smaller number of elements. The finite element model is built by using MSC/Patran (pre-processor and post-processor) and MSC/Nastran (solver).

2.2.1 Meshing

The surface model is generated using points and curves according to the dimensions specified in the geometric model. The software provides automated meshes like Isomesh, which is used for meshing the square and rectangular surfaces of the model and Paver mesh is used to mesh the inclined vertical and horizontal surfaces and trimmed surface with number of holes. The entire model is meshed with a four-noded shell element. In Patran, it is represented as QUAD4. The CQUAD4 is the most commonly used element for modeling plates, shells, and membranes. The CQUAD4 can represent in-plane bending and transverse shear behavior. The CTRIA3 element is a triangular plate connecting three grid points. The CTRIA3 is most commonly used for mesh transitions and filling in irregular boundaries. The element may exhibit excessive stiffness, particularly for membrane strain. Thus, as a matter of good modeling practice, CTRIA3 should be located away from areas of interest whenever possible and model is checked for its quality and connectivity. Quality of surface elements is maintained as per standards. Important quality issues are (1) Aspect Ratio, (2) Jacobian, (3) Skew Angle, (4) Warpage.

After quality checking, assembly is carried out by connecting rigid elements (RBE2), thus the main component of spacecraft and propellant tanks are idealized using CQUAD4 and CBAR elements.

2.2.2 Material Properties

Based on the laminate thickness, central thrust cylinder is divided into six zones and is made up of sandwich construction, aluminum core and CFRP face skin, for propellant tanks titanium alloy material as has been used. CFRP BD lamina is modeled as 2D orthotropic materials. For lamina, Young's modulus in fiber direction, Young's modulus in transverse direction and in-plane Shear modulus are taken into account. A Hoffmann failure criterion is used for these laminae. To implement the failure criterion, tensile and compressive strengths along fiber and transverse directions, in-plane shear strength and bonding shear strength of corresponding laminae are used. These laminae are combined and stacked in a required order using PCOMP material property card. Thickness and orientation of each layer is also mentioned in the PCOMP. These PCOMP material properties are assigned to corresponding shell elements.

2.2.3 Loads & Boundary Conditions

The total spacecraft mass is divided into two parts (1) dry mass of 2800 kg and (2) wet mass of 3200 kg. The dry mass is split and lumped at a distance of 2.5m and 3.5m from base of spacecraft and wet mass is distributed to four propellant tanks. In static analysis, inertia load was applied to determine strain, SR, MOS and Hoffmann failure index.

Spacecraft is connected to launch vehicle by a clamp band and hence the boundary condition is assumed as cantilever, in which both translations (Tx, Ty & Tz) and rotations (Rx, Ry & Rz) are constrained at one end and other end is free. The propellant tanks are placed between free end support panel and fixed end support panel. Tanks are attached to spacecraft with the help of RBE2 elements. The boundary conditions are assumed as fully fixed at tank base(i.e fixed end) and other end(i.e free end) translation (Tz) and rotations (Rx, Ry & Rz) are made free.

2.3 Configuration-1

In first configuration, four propellant tanks are supported by a simple 50 mm core and 0.5 mm face skin honeycomb sandwich panel (fixed end support). This Panel has a central cut out, which is connected to the central thrust cylinder by a ring. Load coming from the tanks is transmitted to central thrust cylinder (Refer figure 2.2).

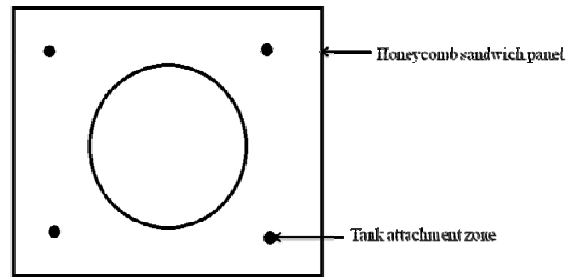


Fig. 2.2 : Top view of fixed end support Honeycomb sandwich panel

2.3.1 Normal Mode Analysis

Normal mode analysis is carried out for every 0.5 mm face skin thickness increment (Refer figure 2.3) and keeping core thickness constant and first 10 natural frequencies and corresponding modal effective masses are extracted.

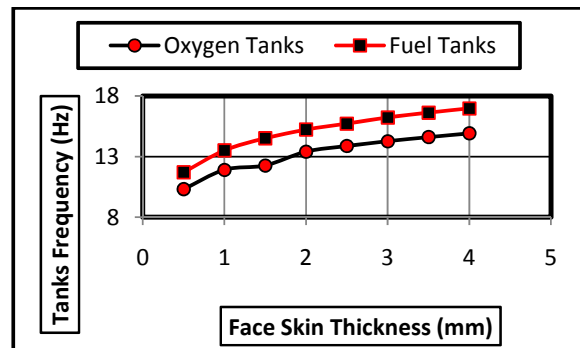


Fig 2.3 Iterative process for Configuration-1

2.3.2 Results and Discussions

In this design, it can be noted that there is no significant variation in frequencies with variation of skin thickness. The achieved frequency for tanks is much lower than the requirement of >27Hz for oxygen tanks and >30Hz for fuel tanks. Hence, this configuration needs to be modified.

2.4 Configuration-2

Configuration-2 has been developed, in which 200 mm×200 mm doublers are embedded into 50 mm core and 0.5 mm face skin honeycomb sandwich panel at tanks attachment zone to increase the local stiffness (Refer figure 2.4).

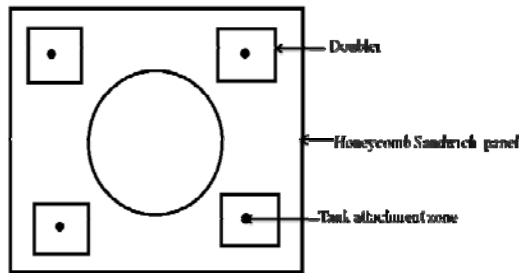


Fig. 2.4 : Top view of fixed end Honeycomb Sandwich panel embedded with doublers

2.4.1 Normal Mode Analysis

Normal mode analysis is carried out for every 1mm skin thickness of doublers (Refer figure 2.5) and first 10 natural frequencies and corresponding modal effective masses are extracted.

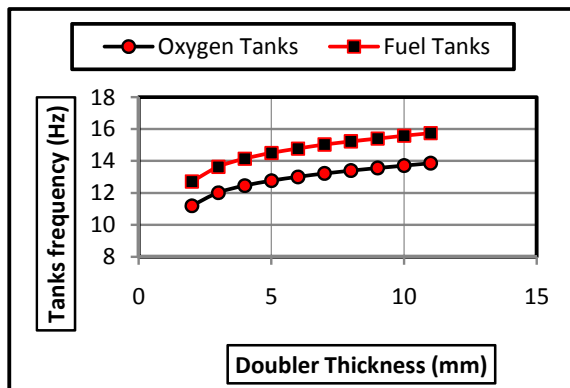


Fig 2.5 : Iterative process for Configuration-2

2.4.2 Results and Discussions

In configuration-2, number of iterations was carried out by varying the doublers skin thickness, which starts from 2 mm (Refer figure 2.5). Even with these modifications, the achieved frequency for tanks is still much lower than the requirement.

2.5 Configuration-3

To improve axial frequency of tank, a hollow rectangular beam of 200 mm×50 mm is embedded inside 50 mm core and 0.5 mm face skin honeycomb sandwich panel (Refer figure 2.6).

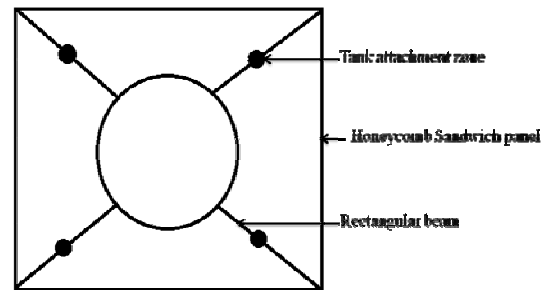


Fig. 2.6 : Top view of fixed end Honeycomb Sandwich panel embedded with Rectangular beam

2.5.1 Normal Mode Analysis

Normal mode analysis is carried out for Configuration- 3 for every 1mm beam thickness increment (Refer figure 2.7) and first 10 natural frequencies and corresponding modal effective masses are extracted.

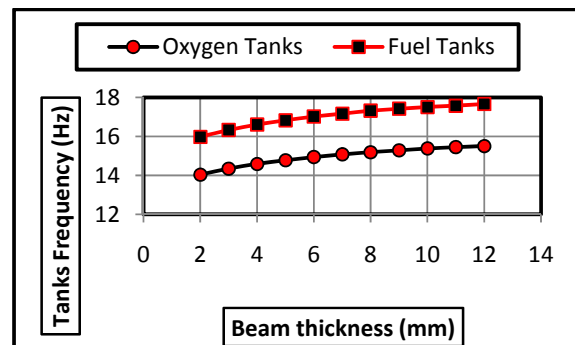


Fig 2.7 : Iterative process for Configuration-3

2.5.2 Results and Discussions

In configuration-3, number of iterations was carried out by varying the thickness of the beam. The initial iteration starts with 2 mm (Refer figure 2.7). There is no significant improvement in the axial stiffness of the tanks

2.6 Configuration-4

In configuration-4, two circular struts are placed below the fixed end deck for each tank and the other end of the strut is connected to cone portion of central thrust cylinder. Here, fixed end panel thickness is as nominal as 50 mm core and 0.5mm face skin (Refer figure 2.8).

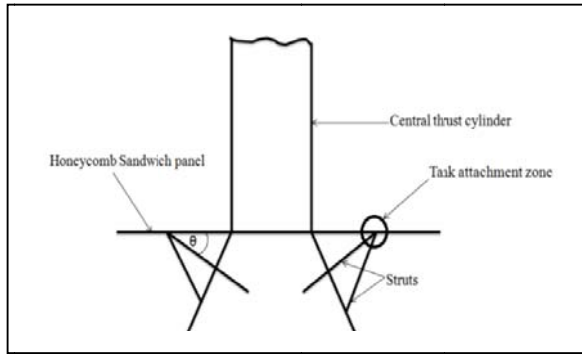


Fig. 2.8 : Front View of fixed end Honeycomb Sandwich Panel with Struts

2.6.1 Normal mode analysis

Normal mode analysis is carried out for configuration-4 by changing the angle, diameter and thickness of the struts (Refer figure 2.9).

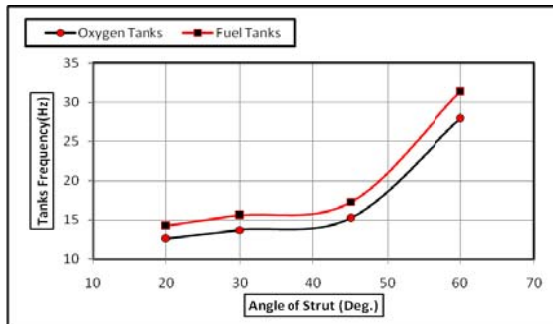


Fig 2.9 : Iterative process for Configuration-4

In configuration-4, the variation of frequencies is very significant with changing angle and diameter as compared to thickness. Finally configuration-4 is meeting stiffness requirements i.e. >27 Hz for oxygen tanks axial mode and >30 Hz for fuel tanks axial mode respectively.

2.6.2 Results and Discussions

In the first three configurations, results were not satisfactory. This in turn led to the development of configuration-4, which is done in unique way by providing struts. Number of iterations was carried out by varying the angle, diameter and thickness of the struts (Refer figure 2.9). It can be noted that the variation in stiffness results is very high, with increase in angle of the struts. For the final configuration, the frequencies were found to be above 27Hz and 30Hz corresponding to oxygen and fuel tanks axial modes. After meeting axial frequencies of propellant tanks, the global frequency was found to be < 8Hz and therefore further iterations were carried out by changing the lay-up of

central thrust cylinder and final design was arrived satisfying all the requirements (Refer figure 2.10).

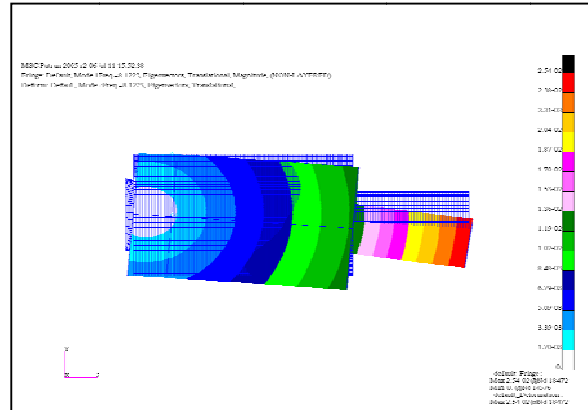


Fig.2.10 (a) : Front view of Global lateral mode of the spacecraft (8.12 Hz)

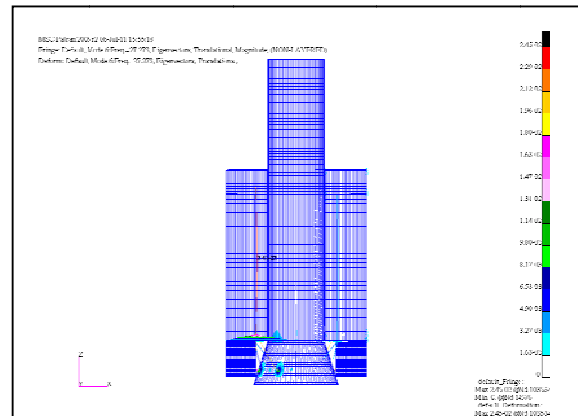


Fig. 2.10 (b) : Front view of Oxygen tanks axial mode (27.2Hz)

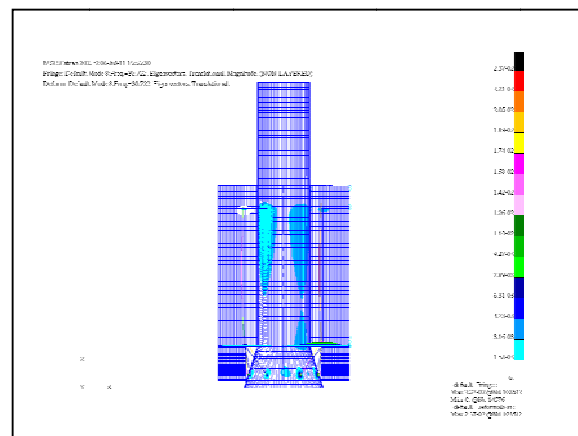


Fig. 2.10 (c) : Front view of Fuel tanks axial mode (30.7Hz)

3. LINEAR STATIC ANALYSIS

In linear static analysis, the inertia loads are applied on the support structure along lateral (X, Y) and longitudinal (Z) directions. The strength of the support structure is determined by failure index using Hoffmann failure criteria. The failure index of the support structure should be less than 1. Hoffman failure criteria take into account the difference in tensile and compressive strengths along longitudinal and transverse directions of the lamina.

3.1 Load Cases

Static analysis is carried out to determine displacement component, strain component, Hoffman failure index, SR and MOS for different load cases as given in the table 3.1

Table 3.1 Load cases

Load case	Lateral(X)	Lateral(Y)	Longitudinal(Z)
LC1		1.25g	-7.5g
LC2	1.25g		-7.5g
LC3		2.5g	-4g
LC4	2.5g		-4g

3.2 Results and Discussions

Static analysis is carried out for optimum design model to determine strain, Hoffmann failure index, SR and MOS for all zones of the central thrust cylinder, for the given load cases are tabulated in tables 3.2 to 3.5. Strain and Hoffmann failure index was found to be maximum at the base of central thrust cylinder and minimum at tip. For load case 4 strain components and Hoffmann failure index were plotted in figures 3.1 and 3.2.

Table 3.6 shows zone wise variation of strain and Hoffmann failure index. Load cases 3 & 4 are critical for design and have sufficient margins of safety.

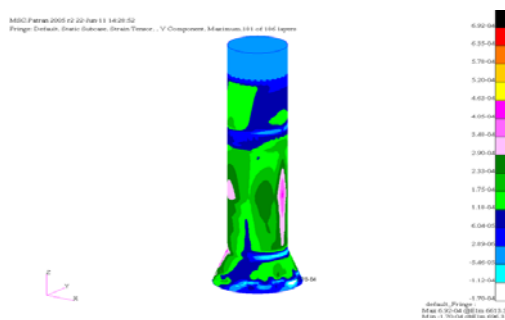


Fig. 3.1: Fiber direction strain plot for LC 4

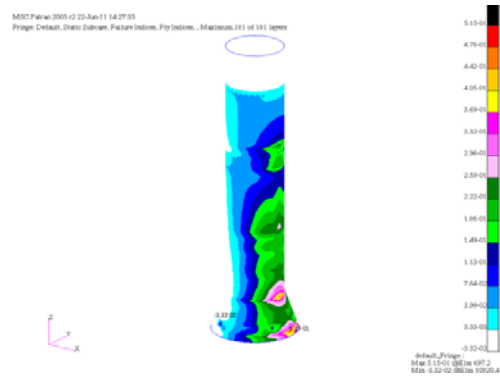


Fig. 3.2 Hoffmann Index plot for LC 4

3.2.1 Stress Analysis for LC 1

Table 3.2 Results of stress analysis for LC 1

Maximum Strain (microns)		Hoffmann Failure index	SR	MOS
X	Y	0.386	2.82	1.82
327	322			

3.2.2 Stress Analysis for LC 2

Table 3.3 Results of stress analysis for LC 2

Maximum Strain (microns)		Hoffmann Failure index	SR	MOS
X	Y	0.386	2.49	1.49
335	310			

3.2.3 Stress Analysis for LC 3

Table 3.4 Results of stress analysis for LC 3

Maximum Strain (microns)		Hoffmann Failure index	SR	MOS
X	Y	0.501	1.55	0.55
779	492			

3.2.4 Stress Analysis for LC 4

Table 3.5 Results of stress analysis for LC 4

Maximum Strain (microns)		Hoffmann Failure index	SR	MOS
X	Y	0.501	1.55	0.55
779	492			

Table 3.6 Variation of strain and Hoffmann failure index for different zone (LC 4)

- To satisfy the requirement of global lateral mode of the spacecraft $>8\text{Hz}$, central thrust cylinder lay-up sequence were tailored until the condition is satisfied.
- Static analysis was carried out for finalized design. Displacement, strain components and Hoffmann failure index were found to be within the allowable limits.

ACKNOWLEDGMENT

The Authors would like to thank Dr. Renji, Group Director, Structures Group, ISRO satellite centre, for his full support to carry out this work successfully.

REFERENCES

- [1] Pascual Julio and Trigo Jesus, "Application of New Technologies for Eurostar Central tube" European space Agency August 2005
- [2] J.M.Autric, D.Catherall and C.Figus, "Design, Development and Validation of the Eurostar 3000 large propellant tanks". European space Agency October 2004
- [3] Vijayakumar S, Basha, SK,J, Srinivasan RK. Design details of CFRP honeycomb sandwich cylinder for I-3000 bus. Structures Group,ISRO,2002
- [4] S. Vijayakumar, "Parametric based design of CFRP honeycomb sandwich cylinder for a spacecraft" Composite Structures. Vol.65 (2004), Pg.No.7-12.
- [5] D. Poomani, R.K. Srinivasan and Sanjay Gupta, "Design, Development and Realization of Carbon Composite Structures for Satellites", INCCOM-7, 2008.
- [6] K.K.Sairajan, P.S.Shammad, Thomas k. Joseph and P.S.Nair, "Optimum Design of a Composite Base Structure of a Spacecraft" ISRO Bangalore August 2005
- [7] Byoung Jung Kim and Dai Gill Lee, Development of a satellite structure with a sandwich T-joint, Composite Structures, Vol 92, 2008, Pg No. 460-468

□□□

4. CONCLUSIONS

- A new concept of support structure for propellant tanks is studied and understood well. This resulted in lowering the center of gravity of spacecraft as compared to conventional spacecraft, due to tank is sitting much closer to the separation plane.
- Studies are carried out for four configurations and optimum design, which meets stiffness requirements, is selected.
- Normal mode analysis of support structure is carried out for all the configurations.
- In configuration-1, face skin thickness increased in the steps of 0.5mm and found that there is no significant improvement in frequency.
- In configurations -2 & 3, doublers and embedded beam, thickness increased in steps of 1mm respectively. In these cases also, there is no significant improvement in frequency.
- In configuration-4 iterations were carried out by varying the angle, diameter and thickness of struts and found that frequency is increasing very significantly by increasing angle, compared to thickness.
- With configurations-4, stiffness requirements of $>27\text{ Hz}$ oxygen tanks axial mode and $>30\text{Hz}$ fuel tanks axial mode were satisfied.