Mechanical Behavior Characterization of Aluminium based Honey Comb Structure by Optimized Modeling and Numerical Simulations

Muhammad Yousuf Ayub, Zeeshan Ahmer, Sohail Riffat Khan , Musaddiq Ali Shah
Satellite Research and Development Centre, Karachi, SRDC-K
SUPARCO Plant, Karachi, Pakistan
Email: yousufayub@yahoo.com , zishahmur@hotmail.com , srkhan1958@yahoo.com , musaddiq52@yahoo.com

Abstract—This research work was carried out to apply the homogenized modeling technique and to investigate the behavior of a prototype aluminium honey comb satellite structure under various kinds of mechanical loadings. It describes the static and dynamic analyses of prototype satellite structure that were carried out in Ansys Workbench V12.0. The structure is composed of aluminium honeycomb sandwich panels, fastened to each other by means of end attachments. It takes into account, the modeling approach, employed for modeling the honeycomb panels and to define contacts between the end attachments and walls of the panels. The structure is of cubical configuration. The structure's FE model consists of panels, end attachments and other masses such as masses of subsystem housings mounted on the structure, modelled as non-structural masses so that the overall mass of the system and its centre of mass are identical to the actual physical structure. The boundary conditions for each of the analyses were finalized, based on the Launch Vehicle specifications. The curves of acceleration (g) v/s frequency for harmonic analysis and PSD acceleration for random vibration analysis were then obtained and the results of stresses and the displacements arising in the structure are discussed.

Keywords— Aluminium honeycomb, Dynamic behaviour, Homogenisation technique, Modelling Approach, Satellite structure.

1 INTRODUCTION

This paper focuses on the dynamic behaviour characterization of honey comb material via computerized simulations, using the homogenization technique followed by the discussion of the analysis results.Various configurations for the design of a satellite structure exist that include cubical configuration, circular configuration, hexagonal configuration etc. The selection of any one of these depends on several factors such as the mission orbit, internal space requirements and so on. Now-a-days, aluminium based honey comb material is frequently being used in the aerospace industry due to its high stiffness to weight ratio, good out-of-plane, in-plane properties and low material and processing cost. However, the use of such composite materials within the aerospace industry requires that such material systems are validated in terms of strict damage tolerance principles [1]. For optimal design, the dynamic and impact behaviour of composite systems needs to be understood. Fastening of honey comb panels is yet another important issue that needs to be addressed as these panels cannot be fastened by conventional means. One method if fastening them is by means of end attachments. One side of the end attachment is bonded through space qualified adhesive to one panel whereas other side of the panel is fastened by means of titanium bolts. Special inserts are designed to allow for the passage of titanium bolt through the panel. The material of the honey comb is AL-6061T6 for the face sheets and Al3003 for the core whereas material of the end attachments is AL-2024T4. The primary structure of the satellite must be stiff enough to sustain the loads of the secondary structure during the launching and service life when it is in the mission orbit, without undergoing any significant deformation [2]. For design and test qualification, the launch environments for the satellite structure are specified in terms of quasi-static, base excited sine vibration, and base excited random vibration testing. Using FEM (Finite Element Method), it is possible to simulate the behaviour of a satellite structure during the qualification tests described above [3]. The accuracy of these results, in turn depends upon the modelling assumptions, model discretization and the application of precise boundary conditions [2]. The analyses are complicated by the fact that honey comb core is an orthotropic material whereas the face sheets are isotropic in nature. A lot of work has been previously carried out on modelling the honey comb effectively with very little or no compromise on the accuracy and yet the model being computationally cheaper. [1, 4] This paper describes the preliminary design of a prototype structure that was designed to withstand the launching loads and whose mission orbit is the LEO orbit. A cubical configuration satellite structure having dimensions as 1000*1000*1369.8 mm^3 is proposed and the design is verified by performing structural analyses that include static and various dynamic analyses. FE modelling approach and the dynamic stress analyses approaches and their application to a sandwich structure are discussed with respect to a compromise on the technical accuracy, computational cost. The main emphasis is on the application and verification of homogenization technique for modelling the honey comb panels under dynamic loading. These analyses have been performed in Ansys Workbench V12.0 which is a very powerful tool for analysing large, complex structures.

2 EXPLANATION OF THE PROPOSED IDEA

Each honey comb panel is modelled using the homogenized approach [4, 5], comprising of three distinct volumes. The upper and the lower volumes (having thickness of 1 mm each) have been assigned the isotropic properties of face sheets of Al6061-T6 whereas the middle volume is assigned the orthotropic properties of the core of honey comb (having thickness of 25.4 mm) of material Al3003. This approach has been adopted by a lot of authors and comparison of simulation and experimental results have proven that this approach quite successfully approximates the behaviour of sandwich honey comb panels under impact loading however this research focuses on the application of homogenization technique to simulate the behaviour of honeycomb panels under dynamic loading via the aid of simulations. Later it is intended to verify the simulation results by performing structural testing. The core is therefore modelled as a solid volume [6]; however equivalent properties have been assigned to it so that mass of that solid and actual porous core is the same. This is in conjunction with the research work already existing in literature [1, 2]. Another reason apart from verifying this approach is the simplicity of the resulting mathematical model and ultimately the requirement of a lesser computational cost. The contact behaviour between each of the layer is defined as "bonded". The aluminium Al2024T4 end attachments are also modelled as a solid volume and bonded contact behaviour is defined between an end attachment and the panel of the wall which are physically bonded via adhesive (space qualified eraldite) to one panel and via titanium bolts to the other panel. Figures1, 2 and 3 given below on the next page depicts the orientation of the honey comb panels and end attachments in the structure. The structure's mass including end attachments is 78.50 Kg. This mass includes the mass of the panels and end attachments and comes from the mass analysis performed in ANSYS, by specifying the material properties i.e. density, ANSYS calculates the mass of each component which is later on added together to get the mass of the entire system. Also this mass is in harmony with the mass budget value that was initially allocated for the prototype structure whereas the auxiliary equipments mass (including subsystem housings and electronic housings) is approximately 133.5 Kg. The mass of housings is also taken from the mass budget document. Each of these equipments is modelled as non-structural masses, which is a usual practice in aerospace industries, and applied over the appropriate area of the structure so that the mass-matrix of the mathematical model remains consistent.

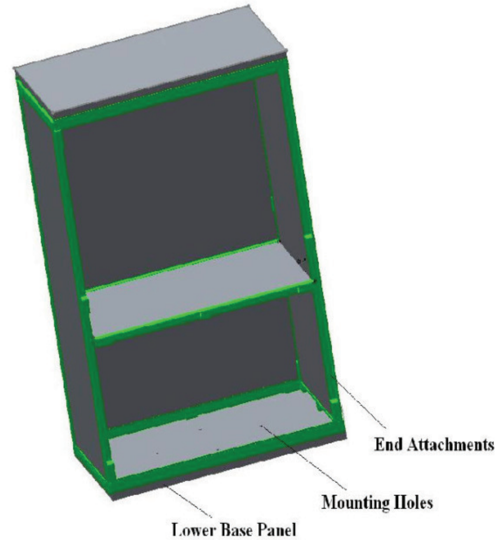

Figure 1 - 3-D model of satellite structure

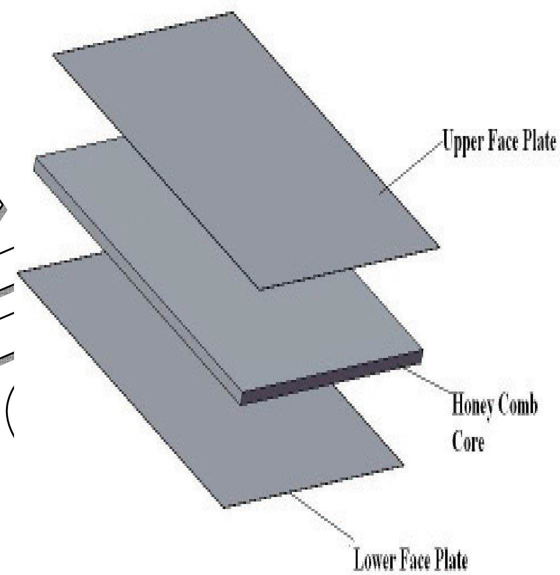

Figure 2 - Detailed view of Honey comb panel

The structure's centre of mass is approximately -697.23 mm in the z-direction and 0.0 in the x- and y- direction respectively from the origin. The structure is mounted to the LVA (Launch Vehicle Adapter) via four mounting holes in the lower base panel which is simulated by modelling four through holes at the exact locations in the lower base panel. The model is meshed with 303036 hex elements and 179570 nodes. [2]

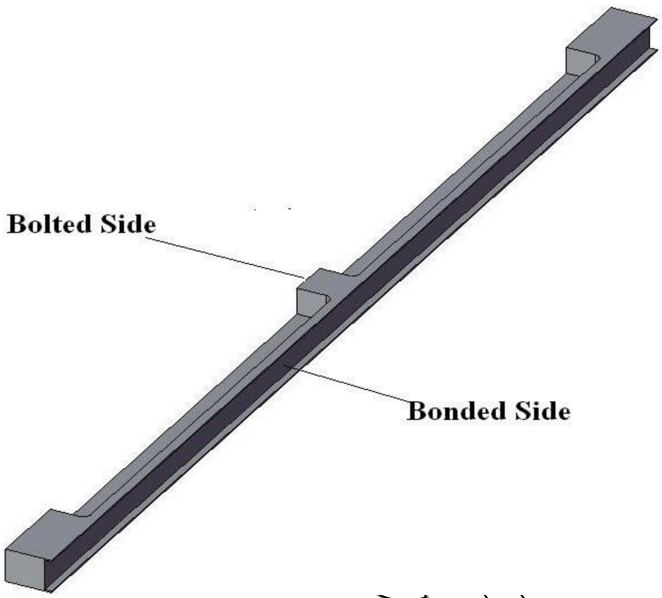

Figure 3 - Detailed view of end attachment

3 NUMERICAL SIMULATIONS

After completing the model, the following analyses were performed:

- Quasi Static Analyses
- Modal Analyses
- Harmonic Analyses
- Random Vibration Analyses

Boundary Conditions and Qualification Levels are as follows:
The structure has to meet the following stiffness and acceleration requirements successfully.

TABLE I - LIMITING MODEL FREQUENCY

1st Natural Frequency of the structure	≥ 30 Hz

TABLE II - ACCELERATION QUALIFICATION LEVEL

Direction	Level
Along satellite Z direction	12g

TABLE III - SINUSOIDAL / RANDOM VIBRATION QUALIFICATION LEVELS

	Normal to the mounting plane		Parallel to the mounting plane	
	Frequency (Hz)	Level (o-p)	Frequency (Hz)	Level (o-p)
Sinusoidal	5 ~ 17	10.3mm	5 ~ 17	10.1mm
	17 ~ 100	12g	17 ~ 100	8g
Random	10 ~ 50	2.55e-4 g2/Hz	10 ~ 50	2.55e-4 g2/Hz
	50 ~ 100	0.1g2/Hz	50 ~ 100	0.1g2/Hz
	100 ~ 200	3.18e-2 g2/Hz	100 ~ 200	3.18e-2 g2/Hz

The boundary conditions are summarized as follows:
For Quasi static analysis all DOF fixed constraint is applied to the mounting holes and "inertial acceleration" is applied to the body in each of the three directions separately.
For all dynamic analyses, the mounting holes are fixed and appropriate loadings as defined above in tables 1-3 were applied. [7, 8, 9, 10]

4 RESULT DISCUSSION

The maximum stress and deformation for the quasi static analyses were found to exist along the satellite z-direction and their values are given below:

TABLE IV- THE RESULTS OF MAXIMUM STRESS AND DISPLACEMENT FOR STATIC ANALYSIS

Load level	The maximum stress and displacement along z-direction	
	Maximum Stress	Maximum Deformation
12g	302.26 MPa	8.5301 mm

The maximum stress was found to exist at the sharp edges/corners at the end attachments whereas maximum stress occurred on the two side panels. This seems logical as the side panels were the largest in length in lateral direction, correspondingly resulting in the maximum stress. The stress values and displacement values are above the allowable limit. This is due to the fact that the said prototype structure was fabricated using one-shot technique and to assess the bonding of Honey Comb Panels with end attachments. The noticeable thing about the stress distribution, as can be seen from the figure below is the fact that in all the regions except the ones mentioned above is in the range of 68 MPa which is satisfactory. The stress arising above in some of the regions is solely due to stress concentrations and to some extent due to material imperfections that exist in almost any material.

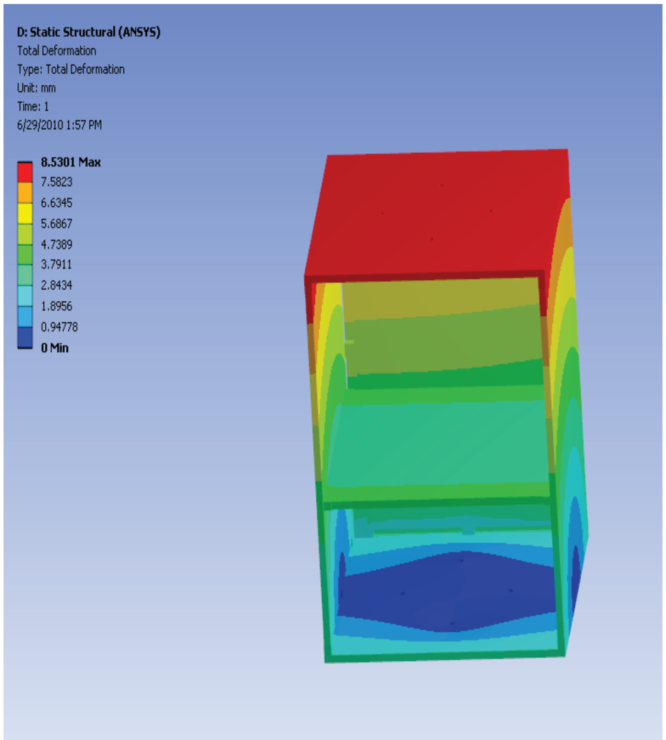

Figure 4 - Maximum static deformation along satellite Z-Direction

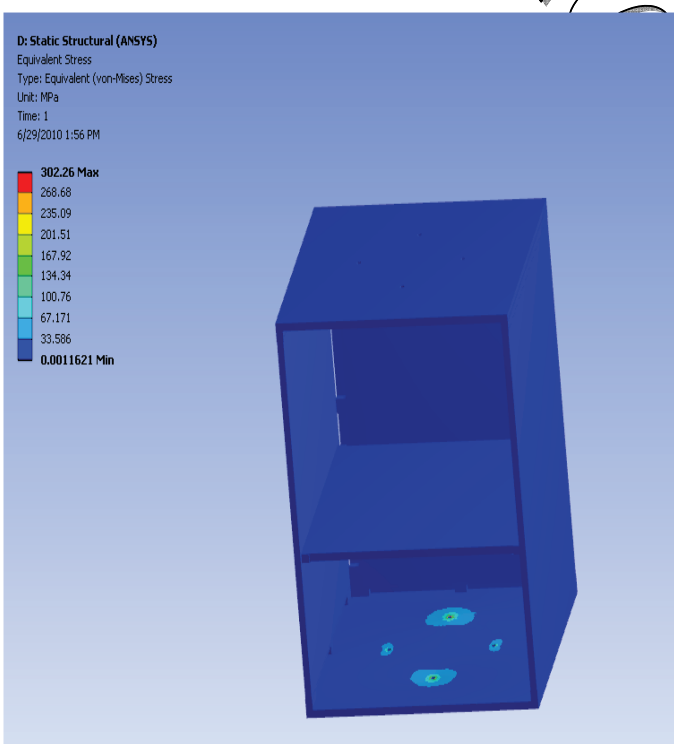

Figure 5 - Maximum static stress along satellite Z-Direction

After running the quasi static analyses, the dynamic analyses that included Modal Analysis, Harmonic Analysis and Random Vibration Analyses were performed. The results of each of these are discussed as follows:
The first five modes of the satellite structure are given below.

TABLE V - FIRST FIVE MODES

Mode	Frequency (Hz)
1.	17.766
2.	19.52
3.	49.781
4.	117.7
5.	156.47

It can be seen that the first two modes of the satellite are below the desired stiffness value. However after the first two modes, the frequency of the 3rd mode and other remaining modes is well above the desired value of 30 Hz [2]. The desired value of stiffness for the first mode can be obtained by increasing the K value of the material which requires thickening of the panels in turn, however it is not a practical solution or by decreasing the mass of the system. The second option is practical and later will be emphasized on, once the CDR (Critical Design Review) is performed. The figures below depict the various mode shapes of the structure.

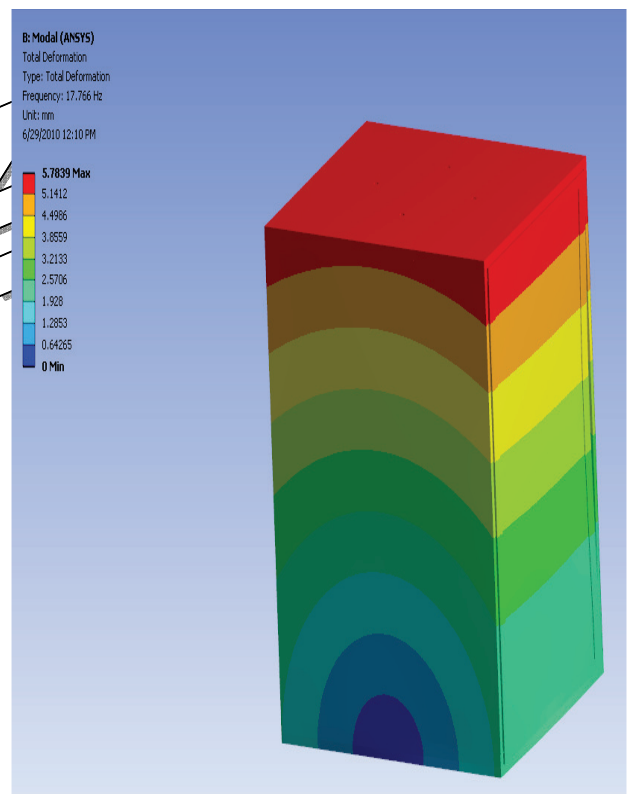

Figure 6 - First mode of the structure

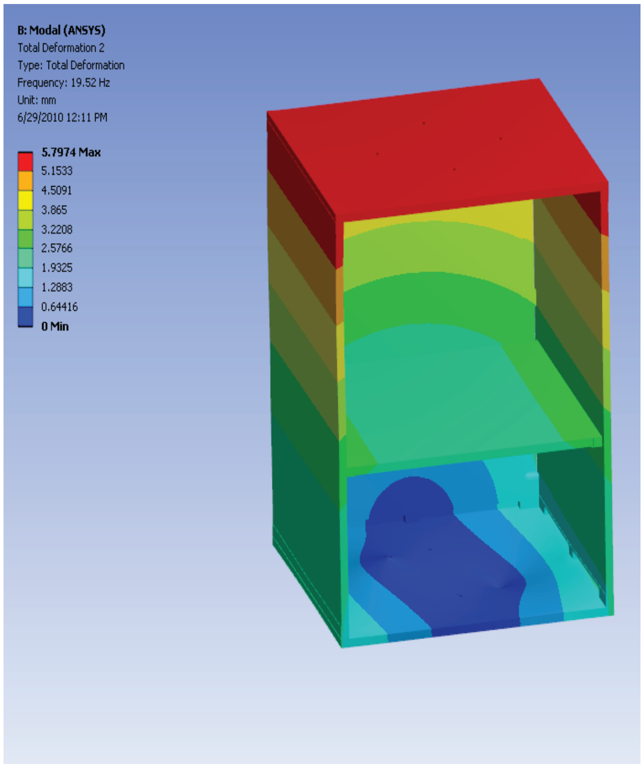

Figure 7 - Second mode of the structure

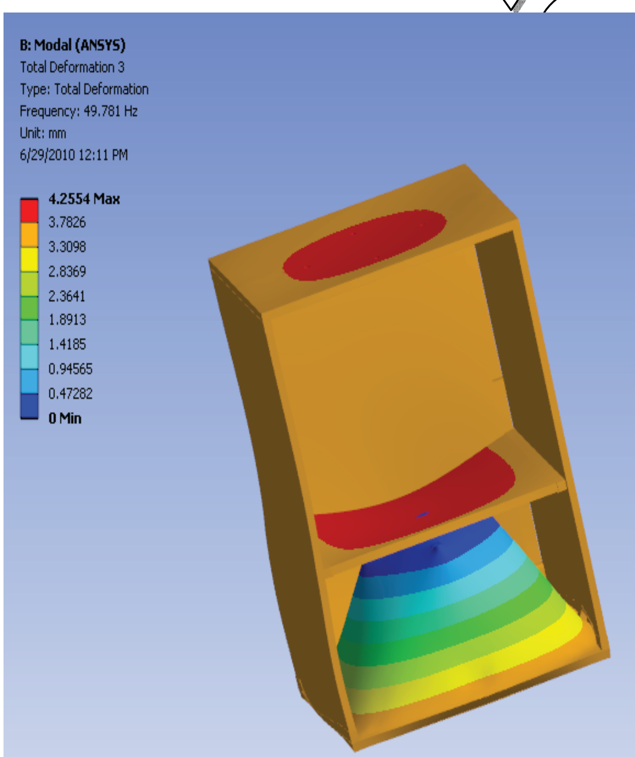

Figure 8 - 3rd mode of the structure

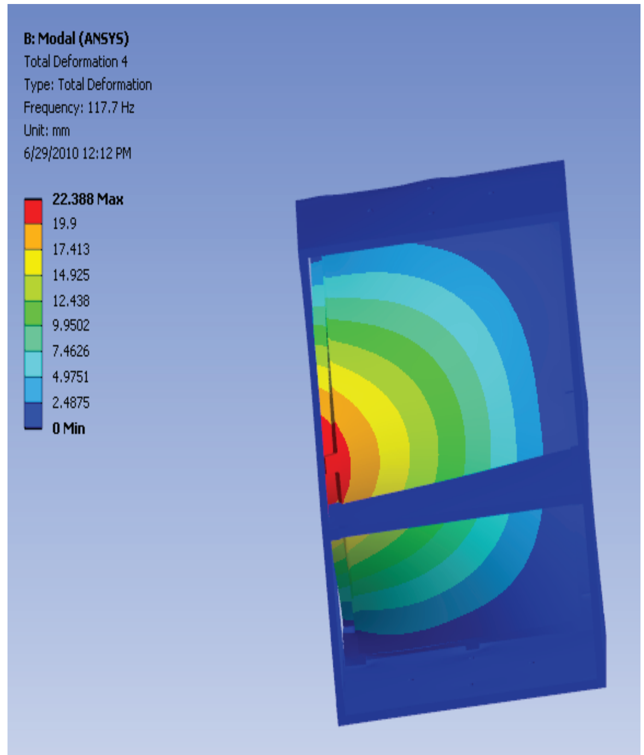

Figure 9 - 4th mode of the structure

Maximum stress and deformation for harmonic analyses was found to exist in the satellite z-direction and at a frequency of 14 Hz. The results are given below:

TABLE VI - RESULTS OF MAXIMUM STRESS, DISPLACEMENT AND ACCELERATION FOR HARMONIC ANALYSES

Response Parameter	Max. Response and Corresponding Frequency
Acceleration (g)	12.389/14 Hz
Stress (MPa)	348.15/14 Hz
Deformation (mm)	35.687/14 Hz

Clearly the stress and deformation values are above the allowable limit. However the acceleration induced in the structure is acceptable. The graph on the next depicts the variation of acceleration (g) v/s Frequency (Hz). It is seen that the structure exhibits peak acceleration at frequency of 14 Hz which is also closest to the natural frequency of the structure. Furthermore it can be observed that the maximum acceleration that occurs in the structure is 12.389 g against the applied value of 12g which is due to the fact that at this frequency, resonance occurs in the structure and as a result the response spectrum reaches its peak which is above the applied value of 12g and it is also at this value that the phase angle between the stress and strain is a maximum and the structure exhibits maximum stress.

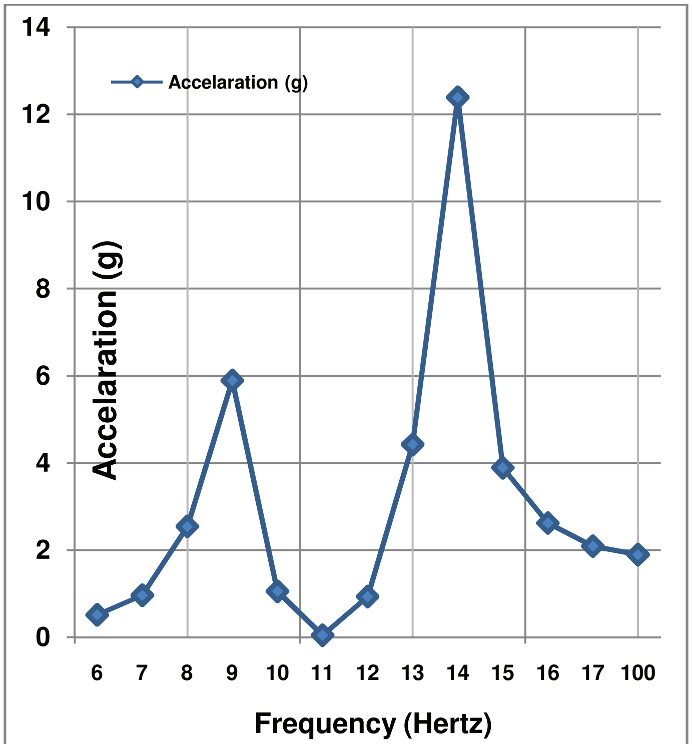

Figure 10 - Curve of acceleration (g) v/s Frequency (Hz) for harmonic analysis

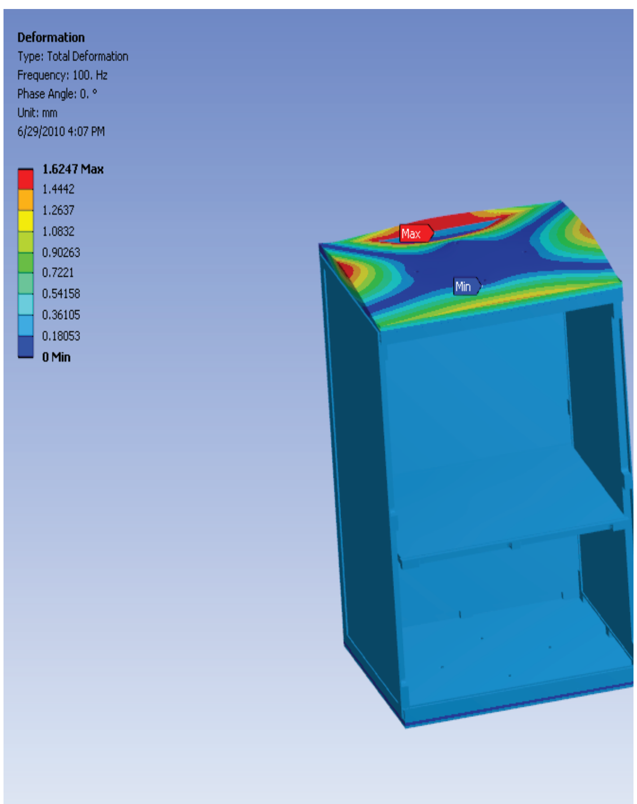

Figure 11 - Deformation along satellite Y-Direction

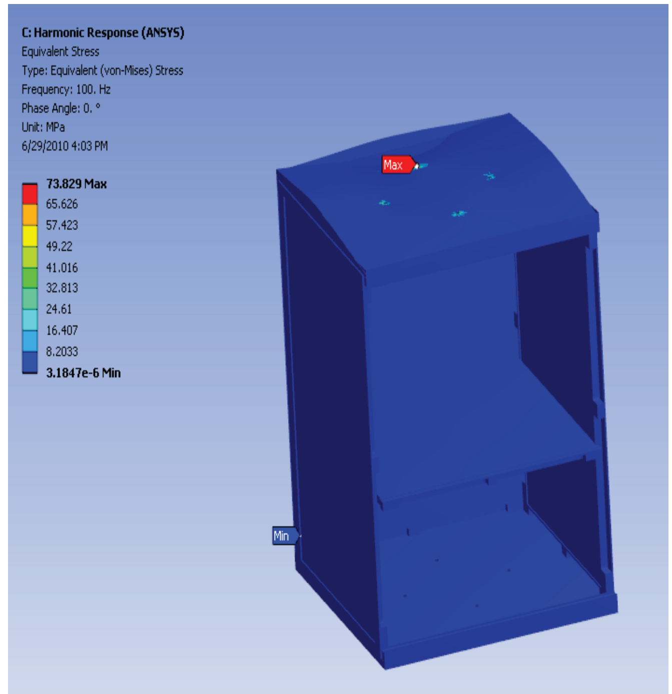

Figure 12 - Harmonic responses at 14Hz

For random vibration analyses, PSD base excitation was applied to the mounting holes in all the three directions separately. The results are discussed below:

TABLE VII - MAXIMUM STRESS AND DISPLACEMENT RESULTS FOR RANDOM VIBRATION ANALYSES

Direction of excitation	Maximum stress	Maximum displacement
X-axis	343.22 MPa	7.61 mm
Y-Axis	1220MPa	10.818 mm
Z-Axis	278.52 MPa	7.3092

It is evident from these results that the values of maximum stress and deformation are greater than the allowable ones. Looking at the curve of PSD acceleration v/s frequency reveals that the resonance occurs continuously in the structure at frequency values between 8-72 Hz and then vibration is damped out by structure sufficiently. This is due to the fact that first 3 modes of the structure lie within the range of 50 Hz and it is between this frequency range that the frequency of the structure co-insides with the frequency of excitation and thus continuously exhibiting resonance which then dies out once the first few natural frequencies have elapsed. The material again exhibits resonance at the frequency range between 170 and 180 Hz but this time the magnitude of acceleration is not so steeper this time due to the fact that one mode lies within this range which is then damped out by the structure.

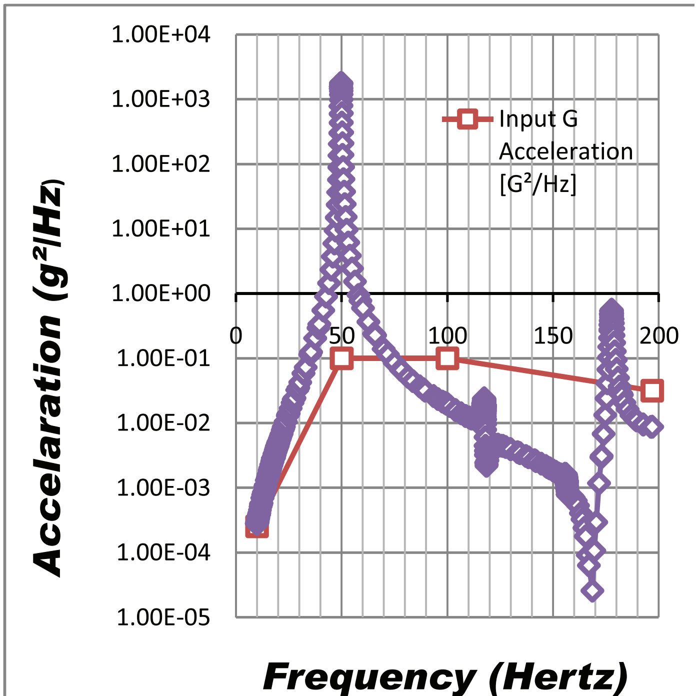

Figure 13 - PSD acceleration (g^2/Hz) Vs Frequency (Hz)

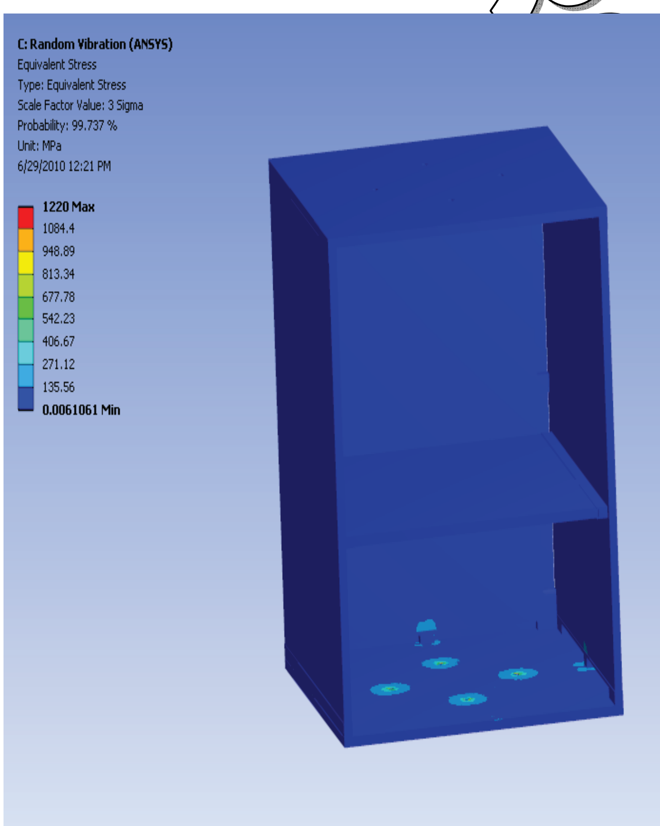

Figure 14 - Maximum stresses along Y-direction for Random Vibration Analysis

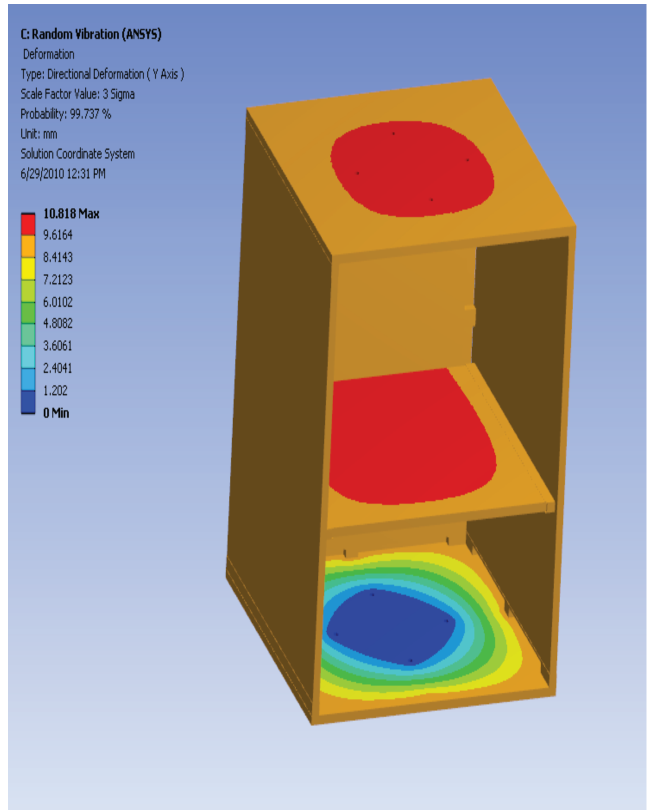

Figure 15 - Maximum deformation along Y-direction for Random Vibration Analysis

5 CONCLUSION

Homogenized modeling approach for Honeycomb was discussed and later applied to a prototype satellite structure composed of Aluminium based honeycomb to characterize the behavior of the composite material under dynamic loading which was accomplished by performing several structural analyses to justify the design of the said prototype structure. The behavior exhibited by the honey comb based structure under static and dynamic loadings seems logical and the analyses results make sense and this certainly conforms to the fact that homogenized approach is quite capable of predicting the behavior of the structure under dynamic loadings. Hence the approach employed for modeling the Honeycomb Panels, modeling end attachments, defining contact behavior between various parts is justified. Later it is intended to verify the simulation results by performing the vibration tests of the said structure on shaker to further ensure the accuracy of the analyses results and to get an insight into the discrepancies that might exist into this approach which will be resolved by improving the mathematical model of the said structure.

ACKNOWLEDGMENT

The authors would like to thank Dr. Muhammad Kashif Khan (Manager SUPARCO, Muhammad Ali Shah (Assistant Manager SUPARCO), Malik Aurangzeb Awan and Waqas Noor (Sub-Engineer SUPARCO) for their guidance and technical support in carrying out this research.

REFERENCES

[1] Davide Caprioli, "FE Simulation of Honeycomb Core Sandwich Panels for the Body", ATZ 01I Volume 110, pp. 26-30, 2008

[2] Ontac, S. Dag, S. Gokler, M.I. Tubitak, Inkara, "Structural Finite Element Analysis of stiffened and honey comb panels of RASAT satellite" in Recent Advances in Space Technologies, pp. 171-175, 14-16 June 2007

[3] K.M. Liew, C.S. Lim, " Analysis of dynamic responses of delaminated honey comb panels", Composite Structures, Volume 39, Issues 1-2, pp. 111-121, Sep-Oct 1997

[4] S. Heimbs, P. Middendorf and M. Maier, "Honeycomb Sandwich Material Modeling for Dynamic Simulations of Aircraft Interior Components" in 9th International LS-DYNA Users Conference on Material Modeling, pp. 20-1 till 20-13, 2006

[5] Kan Zheng, W.H. Liao, Y.T. Qin, "Analysis and Research of Honeycomb Sandwich structure for microsatellite based on equivalent theory", Key Engineering Materials, Volumes 426-427, pages 472-476, January 2010.

[6] Hammad Rehman, Rehan Jamshed, Haris Hameed, Dr. Sajid Raza, " FEA of honey cpmb sandwich panel for continuum properties evaluation and core height influence on the dynamic behavior" in International Bhurban Conference on applied sciences and technology (IBCAST), January 2010

[7] Vyacheslav N. Burlayenko, Tomasz Sadowski, "Finite Element Analysis of dynamic responses for honeycomb sandwich plates initially damaged by skin/core partial debonding" in 8th World Congress on Computational Mechanics (WCCMS), June 30-July 5, 2008, Venus, Italy.

[8] E. Nilsson and A.C. Nilsson, "Prediction and Measurement of some dynamic properties of sandwich structures with Honeycomb and foam cores", Journal of sound and vibration, Volume 251 Issue 3, Pages 409-430, March 2002.

[9] C.W. Schwingshackl, G.S. Agilletic and P.R Cunningham, "Determination of Honeycomb material properties: Existing theories and an alternative dynamic approach", Journal of aerospace engineering ASCE, Page 177-183, July 2006.

[10] Jaroslav Mackerle, "Finite Element Analysis of sandwich structures: a bibliography (1980-2001)", Engineering Computations, Volume 19 Issue 2, page 206-245, 2002.