1. Abstract

This Finite Element (FE) analysis of Central Unit Assembly is simulatedunder different vibration loading conditions. A set of experiments are conducted on random vibrations and shock loads with thickness arrangements. The experimental study together with FE modelling to study its behavior.

Keywords**: Dynamic Analysis,Vibrational Analysis, Hypermesh, Radioss**

2. Introduction

The Central Unit Assembly was subjected to vibrational loads in its operating conditions. So it is necessary to evaluate the displacements and accelerations w.r.t frequencies on the central unit assembly. These vibrational simulations are required to avoid the malfunctioning of the assembly in its actual operating conditions. This electronic assembly consists of many PCB structures, motherboards and other electronic devices which are very small and more important in its operating.

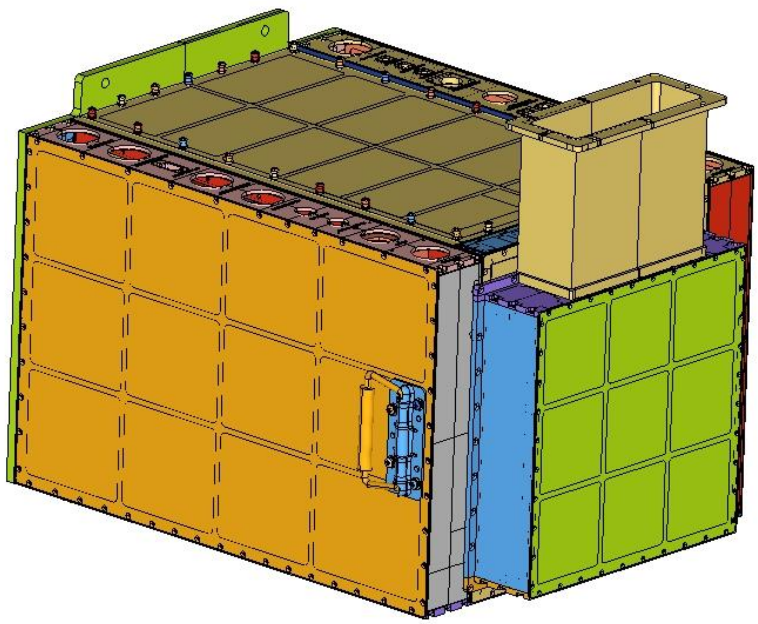

Figure 1: Central Unit Assembly

All fasteners within the assembly are considered for the present analysis and rigids have been used in place of them. A Modal analysis has been carried out to identify its mode shapes w.r.t frequencies before executing the random vibrations and direct frequency response analysis on the central unit assembly. The entire central unit assembly was made of aluminum T6061material. All internal components like PCB's and other thin electronic components are considered as thin shells with respective thickness.

3. Discussion of FEA approaches

The finite element model for Central Unit Assembly was generated using CTETRA10, CQUAD4, CTRIA3 element types. All fasteners have not been considered in the present analysis. PCB's and other electronic components have been considered as just outer shell material. Rest applied as a mass on the individual component to match with the overall mass of the assembly. The load has been given at the constraint location in the assembly. Below figure 2 shows the finite element model and constraint locations in the assembly.

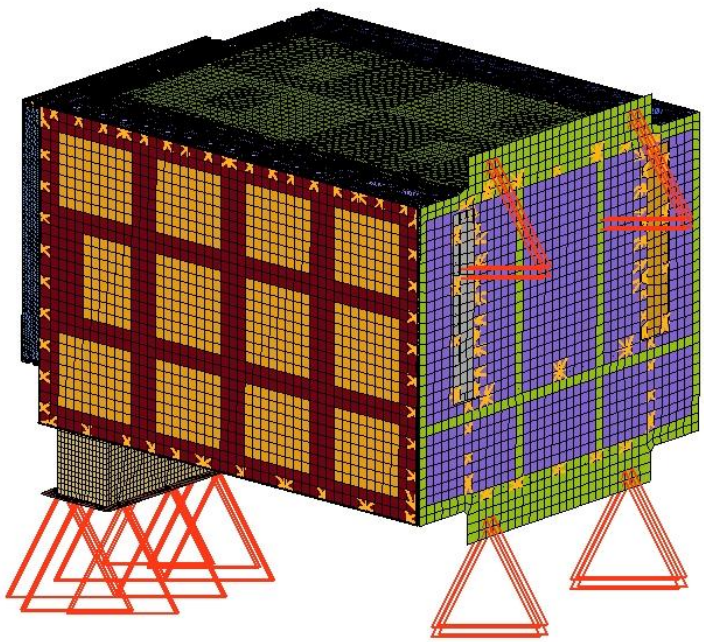

Figure 2: Finite Element Model and Constraints of the Central Unit Assembly

The Load curves with acceleration vs frequency and Acceleration PSD vs. frequency was used as a different loading parameters.

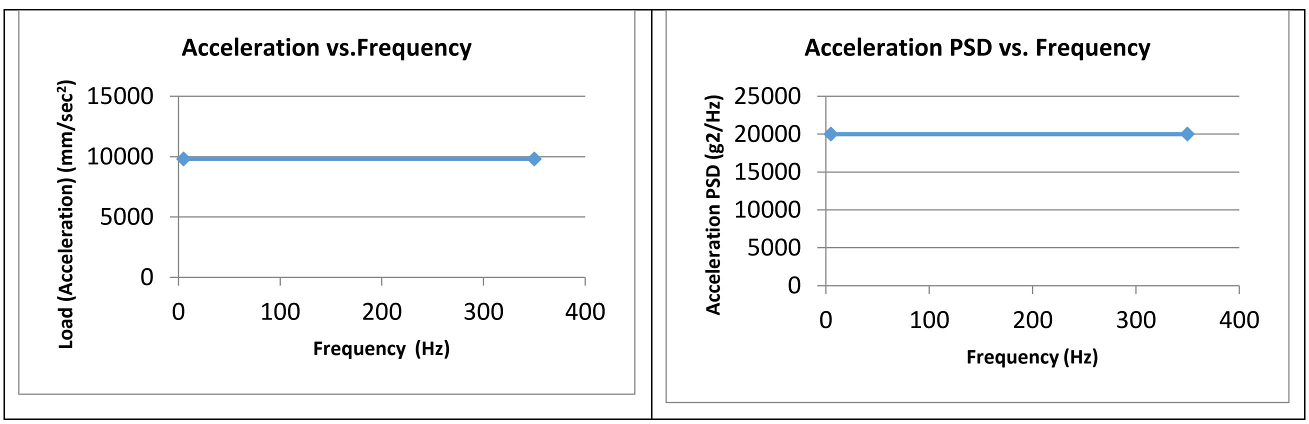

Figure 3: Frequency response loading with 1g and random vibration with ±20g

The acceleration in frequency response was to be simulated in the all three directions, whereas the random vibration analysis was simulated in all six direction (positive and Negative of X, Y, Z directions). The response (output) was chosen with very limited number of nodes in the assembly to reduce the computational time.

4. Numerical Simulation

Finite element simulation of a Central Unit Assembly is carried out using the commercially available finite element package RADIOSS12.0. This solver is capable of performing linear and non-linear stress analysis in a static or dynamic framework.

5. FEA Results

The numerical analysis was conducted using Direct Transient analysis and Modal Transient Analysis.Majority number of output nodes were requested from the outer chassis body which directly impacts the inner electronic components.

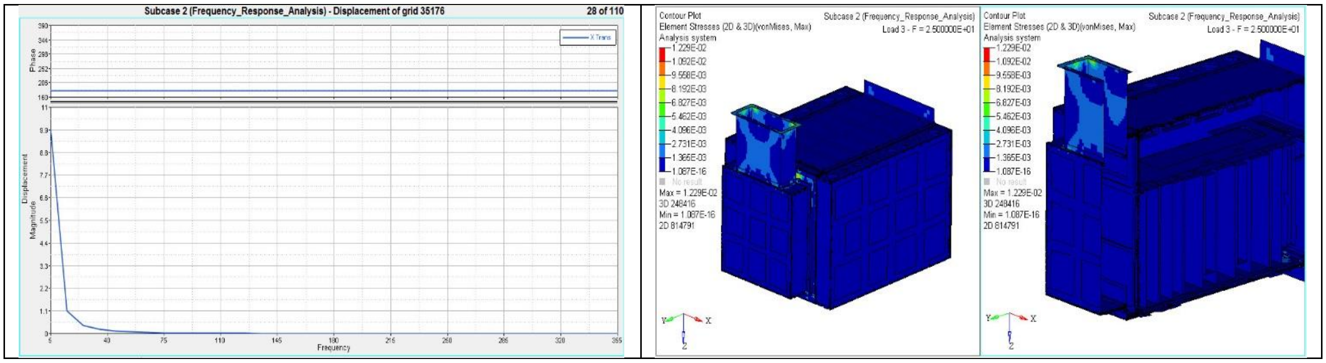

Figure 4: Displacement vs. Frequency and Stress vs. Frequency Contour Plots of a Central Unit Assembly in frequency response analysis in X-direction Excitation

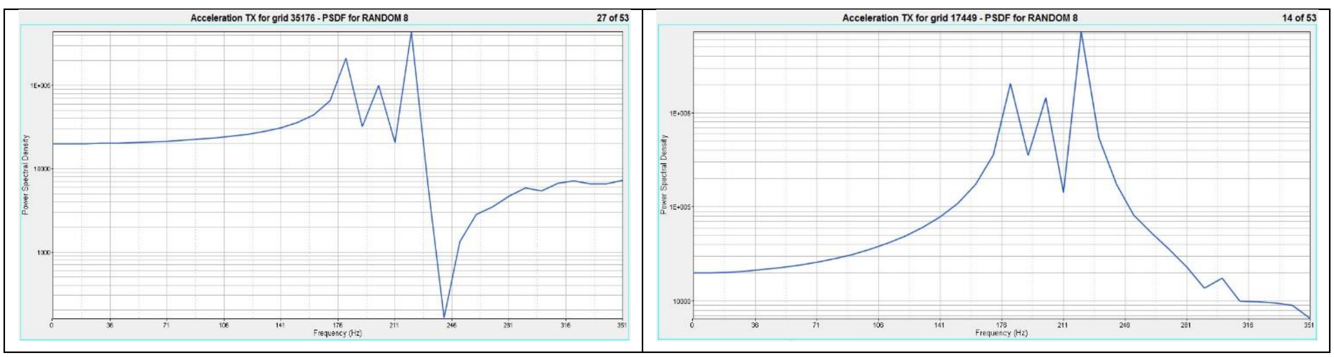

Figure 5: Power Spectral Density vs. Frequency at different Nodes in positive X-Direction Excitation

6. Conclusion

In this paper, an experimental and analytical investigation of central unit assembly that fail due to delamination is presented. FE analysis of stress distribution around the bends is analyzed.

From frequency response analysis the displacement was observed more than 8 mm at before the initiation of first mode. The stress counter at various frequencies was observed as similar and high stresses observed near to constraint locations.

From the random vibrationanalysis the peak accelerations w.r.t frequencies was observed at 230 Hz with more 1E5 g2/Hz in the X-direction excitation. These peak accelerations was observed at the later stages in the Y and Z direction excitations say at 250 Hz and 320Hz respectively.