Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 144 (2016) 576 – 583

Procedia Engineering

www.elsevier.com/locate/procedia

12th International Conference on Vibration Problems, ICOVP 2015

# Vibration Response Prediction of the Printed Circuit Boards using Experimentally Validated Finite Element Model

Somashekar V.N[a], Harikrishnan S[b], PSM Aejaz Ahmed[c], Kamesh D[d]

[a,d] SRG, ISRO Satellite Centre, Bengaluru-560017, India
[b,c]SaiVidya Institute of Technology, Bengaluru-560064, India

**Abstract**

A spacecraft consists of a number of mechanical and electronic sub-assemblies. An electronic package is consists of printed circuit boards placed in mechanical housings which are stacked together. Electronic components are mounted on the printed circuit board (PCB). A spacecraft experiences various types of mechanical loads during its launch such as vibration, acoustic and shock loads. The understanding of dynamic behaviour provides valuable insight to a design engineer, to improve the mechanical design and product reliability when used in harsh vibration condition. Generally electronic package design for vibration loads is verified by conducting tests on actual hardware.

This paper addresses on using basic FEA tool to accurately investigate the dynamic characteristics of the PCB and avoid costly testing methods which require hardware. Here the normal modes & frequency response functions (FRF) of PCB are determined and validated using vibration test on PCB. The validated model is used to predict vibration response for random vibration input. It is shown here how the responses are accurately predicted for random vibration input for a design parameter variation of PCB. The results are also validated using vibration test on PCB.


Peer-review under responsibility of the organizing committee of ICOVP 2015

*Keywords:*Printed Circuit Board (PCB); Spacecraft;Vibration Response.

* Corresponding author. Tel.: +91-9916680738/8951735632
*E*-mail address: krishnan1993@gmail.com, aejazahmed93@gmail.com


Peer-review under responsibility of the organizing committee of ICOVP 2015
doi:10.1016/j.proeng.2016.05.044

## 1. Introduction

A spacecraft experiences various types of dynamic loads during its launch such as vibration, acoustic and shock loads. The electronic packages are designed to withstand the launch vibration environment. The mechanical design of electronicpackages is verified by subjecting to vibration testing. The electronic component failures have been observed whenever there is change in its location on the PCB due to electrical design requirements. The basic failure modes of components mounted on PCB due to random vibration environment are the results of the following conditions: high acceleration levels,high stress levels,large displacement amplitudes andelectrical signals out of tolerance [1]. The design failures during test and evaluation process have resulted in delay in the project schedule.

An effective methodology to verify the design other than testing has to be evolved and finite element (FE) analysis has been an effective tool in predicting results. The FE analysis of electronic packages has a lot difficulties due to many low mass individual elements present in it. Electronic systems are often simulated using simple masses, springs and dampers to estimate the dynamic characteristics of the system. Simple one and two degree of freedom systems are used to approximate the electronic systems. More complicated finite element models of electronic systems are created to study the dynamic characteristics of the system and to estimate the fatigue life of critical components mounted on the PCB. Finite element models can be either simplified or detailed. Detailed finite element models are built by modeling the PCB and the components. However, this approach is rarely used as it is time consuming and expensive. Instead, simplified models of PCB are created where the components geometry is neglected. The component effects are included by increasing the Young's modulus and density of the PCB FE model, so it effectively behaves as if components were present. The simple geometry of the board is modeled and meshed using 2-D finite elements (i.e. by using flat shell elements). Sensitivity analysis of PCB finite element models was carried out by Amy et al. [2]. They determined the factors of safety by using different simplification methods of modeling the PCB.

Printed circuit boards are multi-layered structures with complex material properties that make the simulation of their dynamic behavior complicated and hence to simplify this, a number of parameters affecting the PCB's dynamics have been studied through experimentation. Dynamic responses such as accelerations, strains are to be closely monitored and controlled to achieve test results which are consistent and predict the structural behavior in reality [3]. Printed circuit boards with multiple layers are suitable candidates preferable for high speed and high density applications. In order to design electronic packaging systems for safety and standards criterion, the accurate modeling of components influence on the multi-layered PCB becomes critical and complex due to various reasons. Numerical model using finite element technique was developed to simulate the mechanical behavior of multi-layered PCB and physical tests were conducted to validate the impact performance [4].

In this paper, vibration analysis of a typical PCB used for space applications is carried out. The normal modes & frequency response functions (FRF) of bare PCB (without components) are determined and validated using vibration test on PCB. The validated model is used to predict vibration response for random vibration input. It is shown here how the responses are accurately predicted for random vibration input for a design parameter change of PCB. The results are also validated using vibration test on PCB.

## 2. Vibration Response Prediction of PCB using FEM

In this study, a six layer PCB used for space applications is considered. The PCB is modelled as isotropic plate with equivalent material properties such as Young's modulus, Poisson's ratio and mass density. FEA simulations of PCB dynamics are made using MSC.PATRAN as pre-processor and MSC.NASTRAN as solver for bare PCB and PCBwith components. Finite Element model consists of 3364 quadrilateral shell elements. Young's modulus of elasticity for PCB is obtained as 15 GPa by conducting a three point bending test for three different samples of PCB. The PCB properties are given in Table (1). The FE model of bare PCB is analysed and validated for two different boundary conditions i.e., *i)* PCB fixed/clamped at nine mounting locations shown in Figure (1) and *ii)* PCB with fixed at eight mounting locations shown in Figure (2).