Modelling and simulation of effect of component stiffness on Dynamic behaviour of Printed Circuit Board

M. Mary Thraza\textsuperscript{1}, Somashekar V N\textsuperscript{2}, Jayaraman S\textsuperscript{3}, Manish Trikha\textsuperscript{4}, Kamesh D\textsuperscript{5}, Venkatesh K\textsuperscript{6} and Ravindra M\textsuperscript{7}

\textsuperscript{1}Aurora Scientific & Technological College of Engineering, Hyderabad, India
\textsuperscript{2,3,4,5,6,7}ISRO Satellite Centre, Vimanapura Post, Bangalore-560017, Karnataka, India
sjayaram@isac.gov.in

January 5, 2018

Abstract

A spacecraft consists of a number of electronic packages to meet the functional requirements. An electronic package is generally an assembly of printed circuit boards placed in a mechanical housing. A number of electronic components are mounted on the printed circuit board (PCB). A spacecraft experiences various types of loads during its launch such as vibration, acoustic and shock loads. Prediction of response for printed circuit boards due to vibration loads is important for mechanical design and reliability of electronic packages. The modeling and analysis of printed circuit boards is required for accurate prediction of response due to vibration loads. Vibration analyses of printed circuit boards are carried out using finite element method. The objective of this paper is to predict the vibration response of a printed
circuit board including the effect of component stiffness. Effect of contribution of component stiffness to the dynamic characteristics of PCB assembly is investigated. Modeling and analyses of PCB with components used for space applications is carried out. The analysis results are validated using vibration tests of PCB.

**Key Words:** Printed Circuit Board (PCB), Spacecraft, Vibration analysis, Component stiffness

1 Introduction

A spacecraft experiences various types of loads during its launch such as vibration, acoustic and shock loads. The electronic packages are designed to withstand the launch vibration environment. Electronics packages are subjected to vibration testing to establish adequate margins. Package component failures due to vibration loads have been observed in the past. The four 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 and electrical signals out of tolerance [1].

It is possible to predict the probability of mechanical failure by a two stage Physics of Failure (POF) approach. The first stage of this approach is defined as the response prediction stage. In this stage, vibration response of the board is calculated through a finite element (FE) model of the PCB component system. The second stage relates this calculated response to some pre-determined component failure criteria, to show whether the attached components can withstand this curvature or acceleration.

Sophisticated 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. Pitarresi [3], Pitarresi, et al. [4], and Pitarresi and Primavera [5] provided the solutions for issues encountered in modeling the PCB assembly that includes wide variety of components.

In this paper, modeling and simulation of a typical component mounted on PCB used for space applications is carried out. First, vibration analysis of a bare PCB is carried out using FEM to determine the natural frequencies. The PCB is modeled using shell elements. The FEM model is validated by conducting vibration tests on the PCB and comparing the simulation and test results. Next, static analysis of the component mounted on PCB is carried out to determine the contribution of component stiffness to the PCB. The effect of the component stiffness to the PCB is calculated in terms of stiffness coefficients of the PCB based on this analysis. The stiffness coefficients give the effective stiffness of the PCB that includes the effect of component stiffness. The component is modeled using beam and shell elements. Subsequently, modal analysis and frequency response analysis are carried out for a PCB with components by using the stiffness coefficients derived from the static analysis.

2 Vibration Analysis of a Bare PCB

In this study, a six layer PCB used for space applications is considered. The PCB is modeled as isotropic plate with equivalent material properties such as Young’s modulus, Poisson’s ratio and mass density. Details of the PCB are summarized in Table 1. The PCB is modeled using PATRAN as pre-processor and MSC.NASTRAN is used as solver. The PCB is meshed with 1800 quadrilateral shell
elements with appropriate thickness. Fixed/clamped boundary conditions are applied at nine locations (PCB mounting locations) by arresting six degrees of freedom for the nodes on the boundary of holes in PCB as shown in Figure [1].

Table 1: Details of PCB

| Parameter                | Value         |
|-------------------------|---------------|
| PCB size                | 250x200x2.1 mm|
| Mass of Bare PCB        | 208.4 gm      |
| Youngs Modulus of Bare PCB | 20 GPa      |
| Poissons ratio          | 0.12          |
| Boundary Condition      | Fixed/clamped |

2.1 FE Simulation Model for Bare PCB

Normal mode analyses were conducted on FE model to extract first three fundamental natural frequencies for bare PCB. The calculated first three natural frequencies are 318.7 Hz, 354.1 Hz and 368.1 Hz for bare PCB. Mode shapes corresponding to these frequencies are given in Figures [2] to [4].

Figure 1: FE Model of bare PCB
Figure 2: Mode shape of bare PCB for first frequency
Figure 3: Mode shape of bare PCB for second frequency
Figure 4: Mode shape of bare PCB for third frequency
2.2 Experimental Test Setup for Bare PCB

The vibration test was conducted by mounting the PCB with screws at nine locations on vibration table. The bare PCB mounted on the vibration table is shown in Figure [5]. Accelerometers are mounted at various locations of the PCB to measure the responses. The vibration test was carried out in the vibration test facility consisting of electro-dynamic shaker, control system, signal conditioners and data acquisition system. The frequency response function (FRF) is obtained using an electro-dynamic shaker by conducting a sine sweep test. In sine sweep test, the input acceleration is given to the test specimen using electro-dynamic shaker and the output acceleration at various desired locations of the test specimen is measured using accelerometer. The ratio of output to input acceleration gives the FRF at that location. The experimental frequency response plot for bare PCB at a specific location is shown in Figure [6].

3 FEM Model Validation

The FEM model is validated by comparing the FEM simulation results and the experimental test results. Simulation and test results for fundamental frequencies of the bare PCB are compared in Table 2. The simulation and test results for bare PCB are matching well. Hence the FEM model is validated.
| Mode No. | Simulation Results (Hz) | Test Results (Hz) | % Difference |
|---------|--------------------------|-------------------|--------------|
| 1       | 318.7                    | 311               | -2.47        |
| 2       | 354.1                    | 351               | -0.88        |
| 3       | 368.1                    | 379               | 2.87         |

4 Static Analysis of a PCB with component

Static analysis of the component mounted on PCB is carried out to determine the contribution of component stiffness to the PCB. The effect of the component stiffness to the PCB is calculated in terms of stiffness coefficients of the PCB based on this analysis. A typical component used in space applications is considered. The component consists of casing and the terminals (pins) as shown in Figure [7]. The component is mounted on the PCB by inserting the terminals in plated through holes and then soldering the terminals on PCB. The physical and material properties for the component are given in Table 3 and Table 4. The component casing is modeled using shell elements and terminals using beam elements.

The static analysis is carried out for a standard PCB size used for 3-point bending test. In 3-point bending test, the PCB is simply supported at the ends and the load is transversely applied at the middle of the PCB. First, the deformation is determined at the midpoint for a bare PCB and next for PCB with the component. Figure [8] shows the deformation plot of PCB with the component. The ratio of the deformations for first to second case gives the stiffness coefficient. The stiffness coefficient gives the effective stiffness of the PCB that includes the effect of component stiffness. This effective stiffness of the PCB can be used in local smearing approach at the component footprint location.
Table 3: Details of Component and terminals

| Element                      | Casing | Terminal |
|------------------------------|--------|----------|
| Young's modulus              | 70 GPa | 159 GPa  |
| Density                      | 8070 kg/m³ | 8000 kg/m³ |
| Poisson's ratio              | 0.33   | 0.33     |

Table 4: Properties of casing and terminals

| Element | Casing | Terminal |
|---------|--------|----------|
| Young's modulus | 70 GPa | 159 GPa  |
| Density | 8070 kg/m³ | 8000 kg/m³ |
| Poisson's ratio | 0.33   | 0.33     |

Figure 7: Deformation of PCB with component

Figure 8: Deformation plot of PCB with component

Table 5: Linear Static Analysis of PCB With and Without Component

| Analytical deformation | Bare PCB (without Component) | PCB with Component | Stiffness coefficient |
|------------------------|--------------------------------|--------------------|-----------------------|
| 2.54-004 m             | 2.65-004 m                     | 1.31-004 m         | 2.02                  |

5 Vibration Analysis of a PCB with component

In this section, modal analysis and frequency response analysis are carried out for a PCB with component. The component considered for static analysis is also taken for vibration analysis. The analysis is carried out for two cases. In the first case, the detailed modeling of the component with PCB is carried out. In the second case, the stiffness and mass of the component is simulated locally on the PCB.
5.1 Detailed Modelling of PCB with Component

In this section, detailed modelling of the component is carried out. The component casing is modeled using shell elements and terminals using beam elements. The FE model of PCB with component is shown in Figure [9]. Normal mode analyses were conducted on FE model to extract first three fundamental natural frequencies for PCB with component. The calculated first three natural frequencies are 324.0 Hz, 360.1 Hz and 385.0 Hz for PCB with component. Mode shapes corresponding to these frequencies are given in Figures [10] to [12].

Figure 9: FEM Model of PCB with component

Figure 10: Mode shape of PCB with component for first frequency

Figure 11: Mode shape of PCB with component for second frequency

Figure 12: Mode shape of PCB with component for third frequency

5.2 Modelling of PCB with Component using Local Smearing Approach

In this section, the stiffness and mass of the component is simulated on the PCB using local smearing approach. An example of a locally
smeared FE model of a PCB is shown in Figure 13. The effect of component on PCB is modelled by increasing the stiffness of PCB over the footprint of the component. The PCB density over the component footprint includes the density of the component. The equivalent Youngs Modulus of the PCB at the component footprint is given by Youngs Modulus of the bare PCB times the stiffness coefficient derived from the static analysis. Hence, Youngs Modulus of the PCB at the component footprint = 20*2.02=40.4 GPa.

![Figure 13: Example of a locally smeared FE model of a PCB](image)

![Figure 15: Mode shape of locally smeared PCB second frequency](image)

![Figure 14: Mode shape of locally smeared PCB first frequency](image)

![Figure 16: Mode shape of locally smeared PCB third frequency](image)

### 6 Results and Discussions

In this section, the results of detailed modelling approach and the local smearing approach are compared. The natural frequencies of the PCB for two approaches are compared in Table 6. The results are matching well. The maximum FRF for the second natural frequency at the component location is shown in Figure 17 and Figure 18 and compared in Table 7. These results also show good...
agreement. Hence local smearing approach is also appropriate for determination of natural frequencies and the FRF. This is especially useful for modelling of PCB mounted with number of components. Detailed component modelling, which is time consuming can be avoided. Instead, local smearing approach can be applied based on the stiffness coefficients obtained for the components. The stiffness coefficients can be obtained by simulation or experimentally for different type of components.

Table 6: Comparison of Natural Frequencies of PCB with component for different approaches

| Mode No. | Natural Frequency (Hz) | Detailed modelling | Local smearing |
|----------|------------------------|--------------------|----------------|
| 1        | 328                    | 324                |                |
| 2        | 363                    | 360                |                |
| 3        | 385                    | 385                |                |

Figure 17: FRF Plot at Component Location for detailed modelling approach
Figure 18: FRF Plot at Component Location for local smearing approach

Table 7: Comparison of FRF at component location for different approaches

| Approach             | FRF |
|----------------------|-----|
| Detailed modelling   | 6.0 |
| Local smearing       | 5.9 |

7 Detailed Modelling for Component Stresses / Strains

For determination of stresses/strains for the component or at the PCB-component interface, detailed modelling approach is required. Stresses/strains for a base excitation of 100 m/s² is determined using detailed modelling of the component on the PCB. The strain plots for the PCB and maximum strains at PCB-terminal interface are shown in Figures 17-18. The maximum strains at PCB-terminal interface occur for the outer terminals of the component. The maximum strains at PCB-terminal interface for the first three natural frequencies are shown in Table 7. The strains are maximum for the third natural frequency. The stress plots for the PCB and maximum stresses at PCB-terminal interface are shown in Figures 19-20. The maximum stresses (axial and bending) for component terminal for the first 3 natural frequencies are shown in Table 8. The terminal maximum bending stress occurs for the outer terminal for
the third natural frequency.

![Figure 19: Stain plot for PCB](image1.png)
![Figure 20: Maximum strain at PCB-terminal interface](image2.png)

Table 8: Comparison of maximum strains at PCB-terminal interface for natural frequencies

| Mode No | Natural Frequency (Hz) | Maximum strains at PCB-terminal interface (°) |
|---------|------------------------|-----------------------------------------------|
| 1       | 326                    | 18.6                                          |
| 2       | 361                    | 32.6                                          |
| 3       | 386                    | 47.4                                          |

![Figure 21: Stress plot for PCB](image3.png)
![Figure 22: Maximum stress at PCB-terminal interface](image4.png)

Table 9: Comparison of maximum stresses for component terminal for natural frequencies

| Mode No | Natural Frequency (Hz) | Terminal Maximum Axial Stress (MPa) | Terminal Maximum Bending Stress (MPa) |
|---------|------------------------|------------------------------------|--------------------------------------|
| 1       | 326                    | 1.95                               | 4.44                                 |
| 2       | 361                    | 2.57                               | 7.77                                 |
| 3       | 386                    | 1.93                               | 11.3                                 |
8 Conclusions

Vibration analysis of a printed circuit board is carried out including the effect of component stiffness. For model validation, the FEM simulation results are compared with experimental test results for bare PCB. The vibration analysis of a typical PCB mounted with a component is carried out using two different approaches: detailed component modelling approach and local smearing approach. For local smearing approach, the effect of the component stiffness to the PCB is calculated in terms of stiffness coefficients of the PCB based on static analysis. The results of detailed modelling approach and the local smearing approach are matching well for the natural frequencies and FRF. Hence local smearing approach is appropriate for determination of natural frequencies and the FRF, since detailed component modelling approach is time consuming. Detailed component modelling approach is required for determination of stresses/strains for the component or at the PCB-component interface. The maximum stresses and strains at PCB-terminal interface occur for the outer terminals of the component.

Acknowledgement

The authors would like to thank Shri K. Venkatesh, Group Director, QAG, Md. Khan Assistant Professor, and Dr. M. Ravindra, Deputy Director, RCA for their constant support during this project. This team is also thankful to Mr. E. Dinakaran, Head, and Onboard Computers section, also thankful to Mr. Nawab System Engineer.

References

[1] Steinberg D. S., Vibration Analysis for Electronic Equipment. 3rd ed. New York: John Wiley & Sons Inc., 2000.

[2] Amy R. A., Aglietti G. S., Richardson G., Sensitivity analysis of simplified Printed Circuit Board finite element models, Microelectronics Reliability, Vol. 49, pp. 791-799, 2009.

[3] Pitarresi, J.M., Modeling of Printed Circuit Cards Subject to Vibration, IEEE Proceedings of the Circuits and Systems Conference, New Orleans, LA, May 3-5, pp. 2104-2107, 1990.
[4] Pitarresi J.M., Celetka D., Coldwel R. and Smith D., *The Smeared Properties Approach to FE Vibration Modeling of Printed Circuit Cards*, ASME Journal of Electronics Packaging, Vol.113, pp. 250-257, 1991.

[5] Pitarresi J.M. and Primavera A., *Comparison of Vibration Modeling Techniques for Printed Circuit Cards*, ASME Journal of Electronics Packaging, Vol.113, pp. 378-383, 1992
