Microelectronics Reliability 49 (2009) 791–799

Contents lists available at ScienceDirect

Microelectronics Reliability

journal homepage: www.elsevier.com/locate/microrel

Sensitivity analysis of simplified Printed Circuit Board finite element models

Robin Alastair Amy [a], G.S. Aglietti [a,*], Guy Richardson [b]

[a] Astronautical Research Group, School of Engineering Sciences, University of Southampton, Southampton SO17 1BJ, UK
[b] Surrey Satellite Technology Limited, Guildford, Surrey, UK

ARTICLE INFO

Article history:
Received 8 January 2008
Received in revised form 23 February 2009
Available online 31 May 2009

ABSTRACT

Many items of electronic equipment are subjected to harsh vibrations during their lifetime, these vibrations can damage electronic components and potentially risk total device failure. One approach to assess this risk is to compare the predicted vibration response of the Printed Circuit Board against a vibration level that is experimentally determined to produce component failure. Theoretically the vibration response can be determined using a simplified model of the PCB, where the components are modelled using a "smeared" approach; however, the error due to using such a simplified approach has not yet been defined. This paper shows a process to calculate the errors produced by such simplification techniques and derives factors of safety that can be used for all future vibration response models, using these factors ensures that future predictions do not underestimate the real response. Additionally, the errors depends on several other values besides the simplification technique, namely the Printed Circuit Board properties and the component: type, location and density. To account for these factors the process will use a sensitivity analysis approach to consider many possible design cases, this approach involves the creation of a large number of randomly created cases, all with different input values and giving different factors of safety. In this way the statistics of the factor of safety can be built up, giving much greater confidence in the results and insight into the drivers of the modelling error.

1. Introduction

Shock and vibration loads imposed on a Printed Circuit Board (PCB) cause stresses on the PCB substrate, component packages, component leads and solder joints. These stresses are due to a combination of the bending moments in the PCB and the inertia forces due to component mass and acceleration [1]. In a worst case scenario these stresses may cause one of the following failure modes: PCB delamination, solder joint fracture, lead fracture or component package fracture, if any single one of these modes occurred total failure would very likely ensue. The probability that one of these failures occurs within a given time depends on the package type, PCB properties, and frequency and amplitude of both bending moments and inertial forces.

It is possible to predict the probability of mechanical failure by a two-stage Physics of Failure (PoF) approach [2–4]. The first stage of this process, defined here as the response prediction stage, calculates the vibration response of the board through a finite element (FE) model of the PCB/component system, incorporating various assumptions to simplify the modelling process. The second stage relates this calculated response to some pre-determined component failure criteria, to show whether the *attached components*[1] can withstand this curvature or acceleration. The work presented here is a contribution to the initial response prediction stage of the PoF process and does not consider in detail the failure criteria stage.

A major difficulty with response prediction is that the PCB's vibration response is altered when a component is attached to it, as the components effectively increase the mass and stiffness of the PCB, this is particularly true when heavy or large components are present as these increase the effective mass and stiffness of the PCB the most. The problem can be solved, in theory at least, by building a detailed finite element model of the PCB and components (where each component is modelled in detail as in Fig. 1); however, this approach is rarely used as it requires a long time to build and solve the model. Instead, the standard practice is to create simplified models where the components geometry is not modelled at all. Instead of detailed component models, 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 relative simplicity and speed of these simplified methods has led them to be more favourable than detailed methods.

* Corresponding author. Tel.: +44 2380 592453.
E-mail address: gsa@soton.ac.uk (G.S. Aglietti).

[1] The term *attached component* refers to any combined PCB and component system, with the intention of emphasizing the different mass and stiffness properties exhibited when a component is attached to a PCB.

0026-2714/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.microrel.2009.04.002

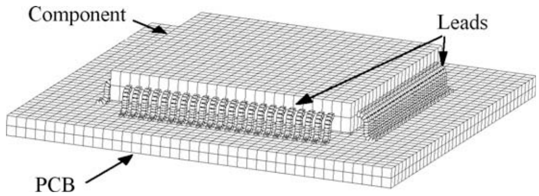

Fig. 1. Example of a detailed FE model of an individual component, a detailed model of a PCB would incorporate several of these and other components over its surface.

This work will build on previous work [5,6] by showing a process to calculate the additional error that is realised when using any one of the several possible simplification techniques. Using a Monte Carlo style sensitivity analysis approach, the calculation will be performed for a variety of different hypothetical configurations to ensure that the results are valid over a greater range of cases than previously possible. This method has already been presented in a previous paper [7], except that the previous work focuses upon the preliminary exploratory stages of choosing the input variables, whilst the current work focuses more upon the creation of safety factors; additionally, the current paper is based upon a more relevant set of input variables than the original, is more practical in nature, and better describes the process used to create the variables. The results of the analysis will be used to create factors of safety; these factors can then be used on subsequent simplified models of any equipment, as long as the equipment falls within the bounds of the different hypothetical configurations that were tested.

2. Simplified PCB FE modelling: current practice

As detailed literature reviews on the entire subject of PCB modelling already exist [8,9], the rest of this section will specifically focus on the area of simplified PCB modelling.

Typically, simplified models of populated PCBs are those where the components' geometry is neglected and only the simple geometry of the board is modelled and meshed using 2-D finite elements (i.e. by using flat "shell" elements). The physical effects of the component are then simulated by increasing the stiffness (i.e. the modulus of elasticity) and the density of the elements of the board, so that these elements behave as if they were carrying a component.

The increase of stiffness can be calculated in one of three ways, either by creating a detailed FEM of an attached component (Fig. 1), by experimental bend testing of attached component specimens [5,6] (Fig. 2), or through analytical methods [10,11]. This process is used not only to calculate the Young's modulus contribution but also the additional shear modulus (see Fig. 3). The results

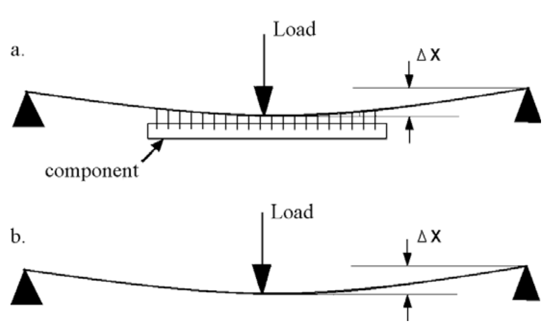

Fig. 2. Method of creating effective stiffness. An *attached component* specimen is either modelled or experimentally tested and the deflection measured (a), a FE model of an unpopulated PCB can be given an artificially high Young's modulus to give the same deflection and therefore effective stiffness as the real specimen (b).

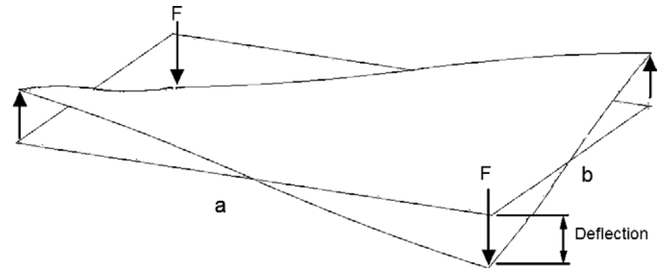

Fig. 3. Method of testing torsional stiffness of a component sample. Loads are placed on diagonal corners and the deflection is measured at the corners.

of these tests are then used to increase the moduli of elasticity of the simplified model, until the model has the same effective stiffness as the attached component, resulting in a patchwork model (Figs. 4 and 5). This increase is only applied at the components' expected location on the simplified model, not over the entire PCB; in this case the model is called "locally smeared".

It is possible to further simplify the modelling process by assuming that the added mass and stiffness can be averaged (or "smeared") over the entire area of the PCB model (see Fig. 5); these models are called "globally smeared". The global stiffness is achieved using an area weighted average of the locally smeared moduli of elasticity, i.e.

$$E_{global} = \frac{\sum E_i A_i}{A_{total}} \tag{1}$$

where E_i is the modulus of elasticity at the location i on the board, and A_i is the surface area of this location, and A_{total} is the total surface area of the board. To smear the mass it is relatively simple: the new density is found from the sum of the components' and PCB mass, which is divided by the volume of the board.

This global smearing technique results in a FE model that does not have a patchwork set of properties to represent each component, but instead has one homogeneous property over the entire area of the model. For this reason, globally smeared models are potentially useful when the final location of the components has not yet been decided.

In addition to locally and globally smeared models, other levels of simplification are also possible, these are just different combinations of smearing either mass or stiffness. These other levels of simplification have been considered in previous work by other authors [5,6] and are as follows:

Simple method: completely neglecting the effect of any components, with the FE model simply reflecting the underlying PCB. The reasoning behind these models is that ignoring the stiffness increases the response (and lowers the natural frequency), whilst ignoring the mass decreases the response (and increases the natural frequency), thus the two compensate for each other.

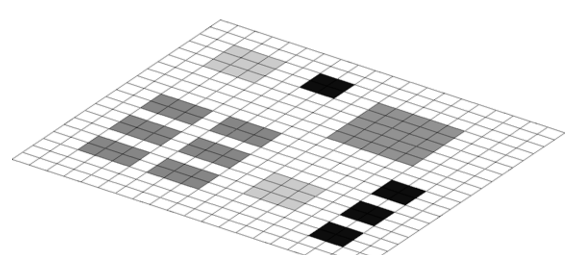

Fig. 4. Example of a locally smeared FE model of a PCB, the shaded locations are intended to model the effects of components and have increased stiffness and density compared to the underlying PCB (non-shaded).