Microelectronics Reliability 91 (2018) 112–119

Contents lists available at ScienceDirect

Microelectronics Reliability

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

Analytical model for the transient analysis of electronic assemblies subjected to impact loading

Mohammad A. Gharaibeh[a,*], Quang T. Su[b], James M. Pitarresi[b]

[a] Mechanical Engineering Department, The Hashemite University, Zarqa 13115, Jordan
[b] Mechanical Engineering Department, Thomas J. Watson School of Engineering and Applied Sciences, Binghamton University, State University of New York, 4400 Vestal Parkway East, Binghamton, NY 13902-6000, United States of America

ABSTRACT

This paper presents an analytical solution for the electronic assembly subjected to shock loading transient analysis problem. A previously published solution is adopted here and hence modified to solve for the transient solution. The results of this solution were thoroughly correlated with measurements and finite element analysis (FEA) data in terms of natural frequencies and mode shapes as well as solder axial deflections. Additionally, this solution was used to calculate axial stresses of the most-critical ball grid array (BGA) solder interconnect. Finally, a comprehensive study was carried out to investigate the effect of the assembly geometric and material parameters as well as loading conditions on the solder stresses and related that to the reliability performance of electronic devices subjected to shock and impact loadings.

1. Introduction

In service life, electronic products are prone to shock or impact loadings and that leads to solder interconnects failures. For this reason, the reliability assessment of electronic assemblies under shock loading has become a major concern in electronics industry. Researchers have discovered and developed several experimental studies, finite element simulations and analytical solutions to evaluate the reliability performance of electronics under shock/impact loadings [1–16]. Globally, Joint Electron Device Engineering Council (JEDEC) provides the standards [17–19] of the reliability tests of electronic packages subjected to mechanical shock environments.

For an electronic assembly under bending, it is strongly believed that the solder failures are due to the flexural differences between the integrated circuit (IC) component and the printed circuit board (PCB) [20–22]. Wong et al. [23,24] used the two elastically-coupled Bernoulli beams approach to assess solder axial stresses due to mechanical shock. The same problem of coupled beams was used to compute solder stresses due to symmetrical static loadings [25] and concentrated forces [26,27] in the cases of full and partial elastic coupling [28].

Tee and his co-workers [29–33] presented various analytical implementations for the finite element simulations of electronic packages under impact. The free fall method [29,30] suggests modeling the drop test experiment entirely. However, the "input-G" and the "input-D" methods recommend to only model the test vehicle while the input accelerations (G) or displacements (D) are to be applied at the boundary conditions, i.e., the mounting screw holes locations [31–33]. Yeh et al. [34] proposed the support excitation approach to analyze the dynamics of the drop experiment. Recently, Gharaibeh et al. [35] developed an analytical solution using Ritz method to study the dynamics of electronic package under base vibration.

This paper adopts the previously derived analytical solution by the authors [35], which was developed for solving the electronic package harmonic vibration problem. In the present work, this analytical solution was modified to obtain the transient solution of electronic packages subjected to mechanical shock problem. The structure of this paper starts by the test assembly configuration followed by the modal analysis experiment and finite element modeling details. The derivation and validation of the transient solution are introduced consequently. Finally, the effect of the geometric and material parameters of the assembly as well as the loading conditions on the electronic system fatigue performance is presented.

2. Test assembly configuration

The test specimen used in this paper consists of a squared PCB of $76.2 \times 76.2 \times 1$ mm^3 dimensions with a centrally-mounted squared $17 \times 17 \times 1$ mm^3 Amkor CABGA electrical component having a full 16×16 area array of eutectic $63Sn37Pb$ BGA solder joints with 280 μm standoff height and 540 μm diameter, as shown in Fig. 1. The solder interconnects are evenly-spaced at a 1 mm pitch.

---

* Corresponding author.

E-mail addresses: mohammada_fa@hu.edu.jo (M.A. Gharaibeh), qsu@binghamton.edu (Q.T. Su), jmp@binghamton.edu (J.M. Pitarresi).

https://doi.org/10.1016/j.microrel.2018.08.009
Received 29 May 2018; Received in revised form 17 August 2018; Accepted 23 August 2018

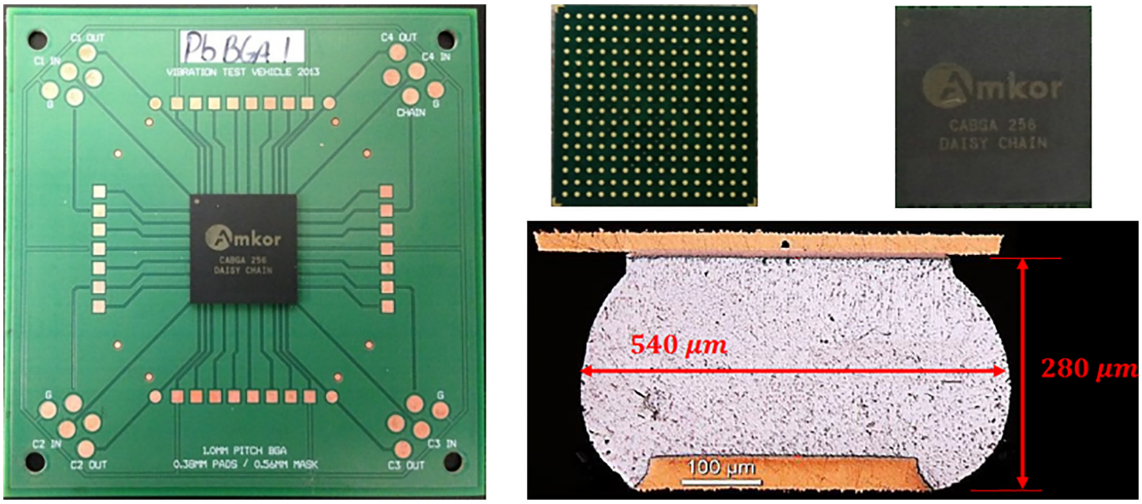

Fig. 1. Test assembly description.

3. Modal analysis experiment setup

The natural frequencies and mode shapes of the present test vehicle are measured using hammer testing experiment, i.e. modal analysis. The setup of this experiment is depicted in Fig. 2. In the setup, the test piece is attached to an Aluminum fixture at the four screw holes. An integrated circuit piezoelectric (ICP) impact hammer was used to gently hit the test sample at certain grid locations. A light-weight accelerometer was placed at a fixed point and used to measure the response acceleration after each hammer impact. The National Instruments data acquisition system model 4413 was used to acquire the acceleration-to-force transfer function for each measurement point. Finally, STAR Modal Version 7.0 [36] was adopted to produce the modal data of the test piece.

4. Finite element modeling

The FE model of the test vehicle used in this paper was built using ANSYS release 19.0 [37]. In this model, a three-dimensional mapped mesh using SOLID185 ANSYS element was generated, as shown in Fig. 3. The natural frequencies and mode shapes of this model were calculated using FEA modal analysis. A mode superposition transient analysis using enforced motion method (EMM), available in ANSYS, was adopted to simulate the test board response due to base excitation shock loading.

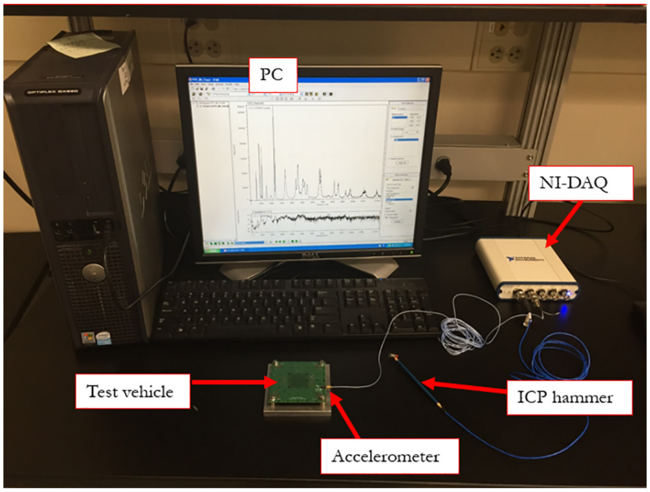

Fig. 2. Modal analysis setup.

In the present FE analysis, the PCB was assumed to have isotropic behavior. Additionally, only linear elastic material properties were considered, as listed in Table 1. The use of linear material properties of the solder alloy might not be entirely true in the impact loading case as solder stresses are expected to exceed the yield stress point. However, this assumption would provide a big pictures about solder stresses and how they are related to the electronic assembly configurations.

The boundary conditions were imposed by restraining the PCB at the top and bottom surfaces of the PCB at the screw holes locations in all directions, i.e., all degrees of freedom were set to zero. The constrained regions on the PCB are equal to those fixed by the screws, standoffs and washers of the modal analysis experiment.

5. Transient analysis analytical solution details

In general, as suggested by Gharaibeh et al. [35], an electronic assembly can be analytically modeled by a two elastically-coupled plates system, depicted in Fig. 4, with the geometric and material parameters listed in Table 2. This analytical model suggests that the PCB is an elastic bottom plate; the component is a rigid top plate and both are elastically connected by individual axial linear springs. Here, the solder interconnects are represented as the axial springs with a stiffness of (K_s) each. In that model and according to Ritz method, the PCB and component displacement solutions are $v(x,y,t)$ and $u(x,y,t)$, respectively. Both solution functions were written in a series form of linear combination of function of space admissible functions of the PCB mode shapes $V_i(x,y)$ or component mode shapes $U_i(x,y)$ multiplied by the time-dependent generalized coordinates $z_i(t)$ as

$$v(x,y,t) = \sum\nolimits_i^N V_i(x,y) z_i(t)$$

$$u(x,y,t) = \sum\nolimits_i^N U_i(x,y) z_i(t) \qquad (1)$$

Gharaibeh's solution only considered first mode analysis, thus $N = 1$. Therefore, the first mode shape of the PCB is:

$$\begin{aligned} V(x,y) = \sum_{n=1}^{N_t} c_{(2n-1)1} sin\left(\frac{(2n-1)\pi x}{l_1}\right) + c_{(2n-1)2} sin\left(\frac{(2n-1)\pi y}{w_1}\right) \\ + c_{(2n-1)3} sin\left(\frac{(2n-1)\pi x}{l_1}\right) sin\left(\frac{(2n-1)\pi y}{w_1}\right) \end{aligned} \qquad (2)$$

Where the coefficients matrix, $c_{(2n-1)m}$, is