Drop Test and Finite Element Analysis of Test Board

Meng-Kao Yeh, Tzu-Heng Huang*

Department of Power Mechanical Engineering, National Tsing Hua University, Hsinchu 30013, Taiwan

Abstract

Most electronic components have high precision and expensive features and people used the drop test to determine the reliability of these electronic components effectively. This study focused on the drop test of FR-4 test board according to the JEDEC standard, to understand the test board's basic mechanical properties and variations of stress and strain on test board. This study used finite element analysis software ANSYS/LS-DYNA to perform a drop test simulation of test board under JESD22-B111 standard, with 0.5ms pulse duration time and 1500G peak acceleration as test conditions. The support excitation method was used to predict the test board response during impact. The results between full model and quarter model were compared to verify the accuracy and efficiency of finite element analysis.

© 2013 The Authors. Published by Elsevier Ltd.
Selection and peer-review under responsibility of the National Tsing Hua University, Department of Power Mechanical Engineering

Keywords: Drop test; Finite element analysis; Support excitation method; FR-4 test board; Explicit analysis

1. Introduction

Since 1980s, electronics industry in Taiwan plays an important role in the global electronics supply chain [1]. With the evolution of electronics industry, consumers demand for higher performance and multi-functions in electronic components and products. As a result, it leads the related manufacturing process to focus on high efficiency and high density features. As the components became smaller, the electronic components directed toward the development of three-dimensional configuration [2, 3], and formed the so-called 3D-IC. As the space utilization of PCB (Print Circuit Board) increases, the product reliability becomes more important. There are many ways to
cause failures in electronic device, among them the most dramatic one is by dropping, especially for portable devices, such like smart phone and laptop. Goyal and Buratynski [4] proposed a method to perform drop test to help the designer estimate the reliability of electronic components in the earlier stage. Nowadays, JEDEC standard is the most-used standard while performing drop test. Since it is hard to keep every experiments under the same conditions, Hokka, et al [5] proposed a method using pneumatic shock impact to improve the drop test instrument.

Owing to many uncertainties during the drop test, people attempt to use computer simulations for more drop test results. Tee et al [6] proposed the input-G method to simplify the whole drop test system into test board and chips only, then built the analysis model using finite element software; after applying varied acceleration on the screw hole, they were able to obtain stress results through the drop test model. Tee and Luan [7] proposed the input-D method, similar to the input-G method, to change the varied acceleration into equivalent varied displacement on the drop test simulation. Yeh and Lai [8] proposed the support excitation method, similar concept as input-G method and input-D method, to fix the screw hole and to apply varied acceleration on the whole model. This method is easier to apply boundary conditions. Wang [9] and Yeh et al [10] used the support excitation method to simulate the drop tests under JEDEC standard.

Since it is not possible to directly obtain reliability results from computer simulation only, in order to find the relation between computer simulation and test reliability results, Darveaux [11] obtained an equation to relate analysis and test results for heat cycle tests using the maximum SED (strain energy density) in ANSYS model. Qu, et al [12] also used Darveaux equation to calculate the drop test reliability and found the simulation results close to the drop test experiment.

In JESD22-B111 standard [13], two types of material, FR-4 substrate and RCC, are suggested for test board. FR-4 is a grade designation assigned to glass-reinforced epoxy laminate sheets, tubes, rods and PCB; while RCC is a material contains only with BT. Since FR-4 PCB, cheaper than RCC, has better mechanical properties and electrical insulating qualities, most people use FR-4 PCB as test board while performing drop tests. In this study, the finite element software was used with explicit method [14] to analyze the test board model under JEDEC standard.

2. Drop Test

Fig. 1 shows the drop test facility under JEDEC standard. While performing drop test, the test board was fixed by locking the screws onto four standoffs; then raised the drop table to certain height for free drop. When the drop table hit the strike surface, an accelerometer, mounted on the base plate, was used to measure the impact acceleration, which was compared with the desired value.

The dimension of test board used in this study is 132 mm x 77 mm x 0.85 mm. According to JESD22-B111 standard [13], the peak acceleration 1500G and duration time 0.5ms condition was used. The drop test experiment was performed to verify the accuracy of drop test simulation by comparing strain variation results.

![Fig. 1 Drop test facility under JEDEC standard.](image-url)
3. Finite Element Analysis

The finite element software ANSYS/LS-DYNA [15] was used to simulate the drop test, and the strain and stress variations of test board was discussed. The Young's modulus of test board was first obtained from tensile test using ASTM D638-10 [16], and then used in the finite element analysis. The finite element method divides the whole model into small elements; every element has nodes with degree of freedom. By using weighted residual methods or calculus of variations, the system simultaneous equations can be obtained and the results can be solved by using numerical analysis [17-19].

3.1. Model construction

In this study the support excitation method [8] was used; by setting reference coordinates on the drop table, the analysis model can be reduced to the test board only, no need to build the whole drop test instrument. Computational time is an important issue for dynamic analysis. To assess the computational time, two models, a full model and a quarter model, were built. On the test board model, the boundary condition were set at screw holes. The size of test board is 132 mm x 77 mm, with thickness 0.85mm, adopted from reference [9, 20].

For explicit analysis with finite element software, SOLID 164 element was used. It is a hexahedral shape element with 8 nodes; each node has three degrees of freedom. The analysis model was partitioned manually and mapped mesh was used to obtain better solutions. Fig. 2 shows the full model and quarter model with mapped mesh for the analysis model of test board.

![Fig. 2 The analysis model for test board with mapped mesh (a) full model (b) quarter model](image)

3.2. Applying boundary condition and loads

When using the support excitation method, boundary condition at 4 screw holes are fixed, as shown in Fig. 3 for the full model and the quarter model. From JESD22-B111 standard [13], peak acceleration 1500G and duration time 0.5ms condition were used. According to previous studies [6, 8-10], varied acceleration load can be approximated to a half sine wave pulse, as shown in Fig. 4. Since the finite element software cannot directly input the half sine wave load in explicit analysis, total 19 time steps to approximate the half sine curve into 18 intervals were applied for the varied acceleration. 4 screw holes D.O.F fixed in all directions  D.O.F fixed in x direction

![Fig. 3 Boundary conditions (a) full model (b) quarter model](image)
4. Results And Discussion

According to Lee, et al [20], the Young's modulus of test board used in this study is 25.4GPa, which is similar to the Young's modulus obtained from tensile test, 26.65GPa, with 4.9% difference. Therefore the Young's modulus and parameters from previous studies [9, 20] were used in the analysis model. Because it was not possible to obtain the stress distribution from drop test experiment, the computer simulation is necessary to obtain the stress distribution in the test board under drop test. To verify the accuracy of finite element model, the strain variation curve is used.

Fig. 5 shows the strain variation curves in the middle area of test board from full model, quarter model and drop test experiment. Table 1 shows the maximum strain values in the middle area of test board from full model, quarter model and drop test. The maximum strain value from drop test is the average of three drop tests. If the chips were connected on test board, the failure more likely to occur when chips facing downward [21]; therefore the strain results were taken from the middle area of the downward surface on test board. From Fig. 5, the trend of the curves for both drop test experiment and simulation results are similar. From Table 1, the maximum strain values in the middle area of test board are close for those from full model, quarter model and drop test experiment; and the maximum strain value of full model is closer to the drop test result than that of quarter model. To use the finite element software ANSYS/LS-DYNA with explicit analysis more economically, the quarter model is recommended for the drop test simulation.

Fig. 6 shows the stress variation curves in the middle area of test board from full model, quarter model. Table 2 shows the maximum stress values in the middle area of test board from full model and quarter model. From stress analysis, Fig. 7 shows the stress distributions when maximum stress occurred in middle area of test board for full model and quarter model. It is obvious that these two stress distribution results are similar. Since the stresses on both sides near the center are low, the chips welded at these two place are least likely to fail under drop test. From Fig. 6 and Table 2, the maximum stress of quarter model is higher than that of full model, probably from the symmetry boundary conditions of quarter model used in drop test analysis. If quarter model is used in future works, this difference should be taken into consideration to correct the simulation results.

<table>
<thead>
<tr>
<th>Table 1. Comparison of maximum strain.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Full model</td>
</tr>
<tr>
<td>Maximum strain</td>
</tr>
<tr>
<td>Error</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2. Comparison of maximum stress</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Full model</td>
</tr>
<tr>
<td>Maximum stress</td>
</tr>
<tr>
<td>Error</td>
</tr>
</tbody>
</table>
4. Conclusions

The finite element analysis was used to simulate the drop test of test board, and the stress distributions at the middle area of test board were obtained. The following conclusions are summarized.

(1) The finite element analysis was verified to effectively simulate the drop test of test board.
(2) Both full model and quarter model were built for drop test of test board using finite element software with explicit analysis; the results obtained from the full model is time-consuming and the quarter model is suggested for future studies.
(3) The maximum stress obtained from the quarter model are greater than that from full model, this difference should be taken into consideration to correct the simulation results.
(4) The stress on both sides near the center are low, chips connected at these two place are least likely to fail under drop test.

5. Acknowledgements

The work was sponsored by the National Science Council, Taiwan, ROC, under the grant NSC 99-2221-E-007-042-MY3 and NSC 102-2221-E-007-030-MY3. The support is greatly acknowledged.
References