Computer Transient Simulation for Dual In-Line Package through Finite Element Analysis and ANASYS Software

Wei Su1,2, *, Wenhao Luo2,3, Xianshan Dong1, Si Chen1 and Agam Tomar4

1Science and Technology on Reliability Physics and Application Technology of Electronic Component Laboratory, China Electronic Product Reliability and Environmental Testing Research Institute, Guangzhou, China
2MOE Key lab of Disaster Forecast and Control in Engineering, Guangzhou, China
3School of Mechanics and Construction Engineering, Jinan University, Guangzhou, China
4Faculty of Engineering and Mathematical Sciences, University of Western Australia

*Corresponding author e-mail: suwei@ceprei.com

Abstract. The reliability of the DIP package is studied by numerical simulation. Firstly, the DIP package, fixture and metal steel are modeled. Based on this, variable thickness papers are laid on the sample. According to the experiment, corresponding excitation and boundary condition are set. Then, the transient analysis result of DIP package is obtained through numerical simulation. The results show the relation of different thickness papers and acceleration-time curve, which provides a beneficial reference for the optimization of DIP packing.

Keywords: DIP; Finite Element; Transient; ANASYS.

1. Introduction
As a simple and common package, DIP (dual in-line package) package is widely used in many small-scale IC packaging processes [1-2]. Especially in the military field, the DIP ceramic package can adapt to the harsh environment, which makes the DIP ceramic package more widely used [3]. With the increase of demand and integration, more and more large-size DIP packages (pin count ≥20) are used for devices. It makes the vibration reliability problem of such devices in harsh environments more and more obvious. Especially when performing random vibration test on large-size DIP ceramic package, the problem of high failure rate cannot be avoided [4-5]. This paper uses finite element analysis software to simulate the DIP package and discuss how to affect the response parameters.

2. Theory of Transient
Integrated circuits will inevitably be impacted in the course of operation. The purpose of transient response analysis is to calculate the response of a structure under time-varying excitation. Transient excitation is defined in the time domain, and the magnitude of each instantaneous excitation is known. Integrated circuits will inevitably be impacted in the course of operation. Its motion differential equation can generally be expressed as:
\[
[M][\ddot{u}] + [C][\dot{u}] + [K][u] = \{F^a\}
\]  

Here \([M], [C], [K]\) represent mass, damping and stiffness matrices respectively, and \(\{F^a\}\) is an external excitation matrix.

The solving methods of Equation (1) can be divided into direct integration method and modal superposition method. Direct integration needs to be solved in each time step. Here, one of the commonly used direct integration methods-Newmark method is given, in a time step:

\[
\begin{align*}
\{\ddot{u}_{n+1}\} &= \{\ddot{u}_n\} + [(1 - \delta)(\ddot{u}_n) + \delta(\ddot{u}_{n+1})]\Delta t \\
\{u_{n+1}\} &= \{u_n\} + \{u_n\}\Delta t + \left[\frac{\alpha}{2} - \alpha\right]\{\ddot{u}_n\} \Delta t^2
\end{align*}
\]

Where \(\alpha\) and \(\delta\) are Newmark integral parameters.

3. Finite Element Model

3.1. Modeling

The CAD model shown in the Fig.1 is established by the external software Solidworks. It mainly consists of three parts: 6061 aluminum alloy fixture, ceramic panel and metal sheet made of 4J29 alloy.

![Fig. 1. Transient analysis DIP simplified model](image)

Different working conditions are set up according to the feasibility of the test, the number of paper at the bottom is one piece. The finite element test is performed by changing the number of top paper, as shown in the TABLE1.

| DIP    | Thickness(mm) | Bottom(layer) | Top(layer) |
|--------|---------------|---------------|------------|
| 0.1+0.54 | 1             | 5             |
| 0.1+1.04 | 1             | 10            |
| 0.1+1.54 | 1             | 15            |
| 0.1+2.54 | 1             | 25            |

When calculating, the paper and the model are seen as a whole. The section diagram shows that the grey part is the paper placed, as shown in the Fig.2:
3.2. Material properties

Material properties are gotten by consulting reference materials and set parameters in ANSYS as Table 2:

| Material             | Elastic modulus (Mpa) | Density (kg/m^3) | Poisson's ratio |
|----------------------|-----------------------|------------------|-----------------
| 6061 Al-alloy        | 1000                  | 700              | 0.3            |
| Paper                | 70000                 | 2700             | 0.33           |
| Ceramics             | 120000                | 7500             | 0.24           |
| 4J29 Kovar alloy     | 210000                | 8100             | 0.3            |

3.3. Meshing

Solid187 contains 10 nodes, and it's a high-order 3D unit. So solid187 element is used for the transient analysis to ensure that the calculation accuracy can meet the requirements [6]. According to the calculation of the grid convergence test, the grid size of the fixture is 3 mm, while the other parts are set to 0.5 mm, as shown in Fig. 3 and Fig. 4:

3.4. Boundary condition

In practice, the fixture is placed on the vibration test bench, and the direction of excitation load is perpendicular to the vibration table. Hence, in the analysis, it is necessary to release the fixture vertical
direction of freedom, that is, the Z direction in this report. Other directions are fixed constraints, as shown in Fig. 5-6.

![Fig. 5. The definition of the boundary conditions](image)

3.5. **Input condition**
The impact function curve of transient analysis is set up according to the test conditions. The curve of shock is half sine wave. Its peak of acceleration is 1500g, and the pulse width is 0.5ms. The shock curve is shown in Fig. 7.

![Fig. 7. Input shock response spectrum](image)

3.6. **Analysis process**
Transient analysis step:
1) Modeling of DIP
2) Applied acceleration-impact curve
3) Solve
4) Analyze results
Fig. 8. Analytical procedures

4. Transient Analysis Result
After the pre-processing was completed, the transient analysis was started to calculate the corresponding acceleration results at each moment. The paper quantity was changed according to the working condition, and all results were aggregated to verify the effectiveness of the method. The acceleration results are shown in TABLE3 and Fig. 9.

Table 3. Data of timedomain analysis

| Time(s) | 0.1mm+0.51mm | 0.1mm+0.91mm | 0.1mm+1.51mm | 0.1mm+2.51mm |
|---------|---------------|---------------|---------------|---------------|
| 5.00E-05 | 2724.4        | 2722.5        | 2725.1        | 2725.0        |
| 1.00E-04 | 6753.2        | 6754.4        | 6755.7        | 6755.9        |
| 1.50E-04 | 10231         | 10223         | 10234         | 10234         |
| 2.00E-04 | 12899         | 12850         | 12899         | 12894         |
| 2.50E-04 | 14285         | 14288         | 14289         | 14288         |
| 3.00E-04 | 14287         | 14284         | 14284         | 14289         |
| 3.50E-04 | 12888         | 12887         | 12889         | 12884         |
| 4.00E-04 | 10224         | 10224         | 10224         | 10224         |
| 4.50E-04 | 6557.5        | 6554.7        | 6557.6        | 6557.7        |
| 5.00E-04 | 2253.4        | 2258.5        | 2253.4        | 2253.3        |

Fig. 9. Visualization image

Finally, the stress distribution of each working condition was calculated to find the weak position intuitively, as shown in the Fig.10 and Fig.11.
Fig. 10. Displacement cloud picture of transient analysis
Fig. 11. Von-Mises stress of transient result

5. Conclusion
This paper uses ANSYS Workbench to simulate how different paper thicknesses affect the corresponding response curves. Firstly, modal analysis is carried out and the correct modal solution of the whole fixture is given. Based on the modal solution, the time - acceleration curve of BGA sample is obtained by harmonic response analysis. The results show that the thickness of paper has little effect on
the response. Therefore, it is not recommended to use this method to change dynamic response parameters.

**Acknowledge**
This work was financially supported by the MOE Key lab of Disaster Forecast and Control in Engineering (20200904010), Jinan University.

**Reference**
[1] Su Fei, Wang Guang-zhou, Yang Hui-ping. Stress evaluation of dual in-line devices in welding process [J]. Journal of Experimental Mechanics, 2010, 25(6):619-624.
[2] Cu Wei-hua, Test method research on reliability enhancement testing for module-level electronic products [D]. Changsha, National University of Defense Technology, 2003.
[3] Zhou Jun, Hao Wei-na, Chai Guo-zhong. Actuality and development of fatigue models of solder joints in micro -electronic packaging [J]. Machinery Design & Manufacture, 2007 (1): 76-78.
[4] M.Pei and J.Qu.Constitutive Modeling of lead-free solders [J]. Advances in Electronic Packaging 2005, IEEE, San Francisco, CA, Jul. 2005: 1307-1311.
[5] John H. Lau, S.-W. Ricky Lee, Chris Chang. Solder Joint Reliability of Wafer Level Chip Scale Packages (WLCSP): A Time-Temperature Dependent Creep Analysis [J]. Journal of Electronic Packaging, 2000: 311-316.
[6] John H. Lau, S.-W. Ricky Lee. Modeling and analysis of 96.5Sn-3.5Ag lead-free solder joints of wafer level chip scale package on buildup microvia printed circuit board [J]. IEEE Transactions on electronics packaging manufacturing, 2002: 51-58.