Study on MEMS board-level package reliability under high-G impact

Jiuzheng Cui, Bo Sun, Qiang Feng, ShengKui Zeng

School of Reliability and Systems Engineering, Beihang University, 37 Xueyuan Road, Beijing, China
cuijiuzheng@ste.buaa.edu.cn
sunbo@buaa.edu.cn

ABSTRACT

Under high-G (10^4 g or above) impact load conditions, the reliability of micro electro mechanical systems (MEMS) board-level package and interconnection are critical concerned that influence the mission success of total projectile. This paper conducts a research on this problem to analyze package reliability using finite element modelling (FEM) and simulation method. Theoretical analysis and mathematical model for failure mechanism of MEMS package under high-G impact are conducted and established. A FEM dynamic analysis is conducted on a typical MEMS board-level leadless chip carrier (LCC) package. Results show that the solder joints are one of the key weakness points that influence the reliability of MEMS package. The maximum effective stress in the structure occurs at the outer corner in the outermost solder point, and the alloy cover and printed circuit board (PCB) have a greater deformation.*

1. INTRODUCTION

In gun-shooting and projectile process, the projectile and its inner components (such as MEMS gyroscope, accelerator, and other electrical components) are suffering large inner pressure and high acceleration load. This type of load features as an extremely peak acceleration (10^4g level or above) and duration of extremely short time (such as 10ms) (Lou *et.al.*, 2005), (Vinod *et.al.*, 2008), (Jiang *et.al.*, 2004). Under this load conditions requirement, it's difficult to design and manufacture a reliable fine component used in projectile. The failure of component is frequently found in this type of usage environment that will influence the total projectile reliability.

MEMS and electronics component board level package and interconnections (solder joints) are key weakness points that influence reliability. While LCC (Leadless Chip Carrier) is the general package type that adopted in MEMS and other electronics with its remarkable advantage of small scale and lower cost (Wei *et.al.*, 2004), (Thomas *et.al.*, 2008). LCC package technique belongs to SMT (Surface Mount Technology), and its reliability problem has got more and more attention. Surface mounted technology (SMT) is widely used in MEMS package, as the solder joints of SMT are significantly small (typically a few millimeters), which turns the solder joints between the printed circuit board (PCB) and MEMS chip into the most weak component during the impact(Tee *et.al.*, 2004).

In present literature and report, the research activities have been focusing on the reliability of board level interconnections in drop impact (with 10^2~10^3g level acceleration) for portable electronics (JEDEC Standard, 2003), (Younis *et.al.*, 2007), covering experimental work together with analytical and numerical modeling studies (Sirkar and Senturia, 2002), (Suhir and Burke, 2000), (Yu et.al., 2003), (Tee et.al., 2004). But the fundamental understanding of the reliability of board level interconnections to high-g impact (with 10^5 g level acceleration) remains limited and is a subject which is still need further studied. Also in these drop impact studies, the analysis is especially focus on all kinds of BGA (Ball Grid Array) packages used in portable electronics (Yu et.al., 2003), (Tao et.al., 2006). While the study on the LCC package reliability generally used in MEMS is not found yet.

This paper provides a research on the reliability of solder joints / interconnections in high-g impact and deal with the dynamics of board level impact and package reliability assessment using finite element modeling and simulation method. Theoretical analysis of failure mechanism of MEMS package is conducted to provide a physical explanation and a FEM dynamic analysis is conducted on a typical LCC MEMS package. Efforts will provide reference for development and practical utilization of MEMS components.

2. THEORETICAL ANALYSIS OF FAILURE MECHANISM

Numerical simulation and experimental validation study have confirmed differential flexing between the PCB and the component packages as the primary driver for the failure of board level interconnections during high-g impact (Wong, 2006). The interconnections (solder joints) of MEMS device and electronics component are key weakness that influences the reliability of MEMS, for the differential deflection between the PCB and MEMS device together with mechanical resonance (Tee et.al., 2003).

Generally, the devices such as MEMS components can be taken as rigid body compared to PCB, bending moment will be introduced in PCB during high impact, which will make the PCB produce flexure deformation. The flexure deformation will introduce repeated pressure and compressive stress to the interconnections (solder joints) between MEMS device and PCB, which will result in connection failure of solder joints. The smaller the deformation is, the smaller the stress and strain are, and the time to recover to static equilibrium (the oscillation of impact) is shorter, which can increase the reliability of solder joints.

The PCB are fixed to the base by bolt, considering the transfer process of stress wave, the dynamic response of MEMS package during high-g impact can be simplified to a board supported by each side and the load can be simplified to uniform load applied to underside of PCB, the two-dimensional simplified universal model is shown in Fig.1.

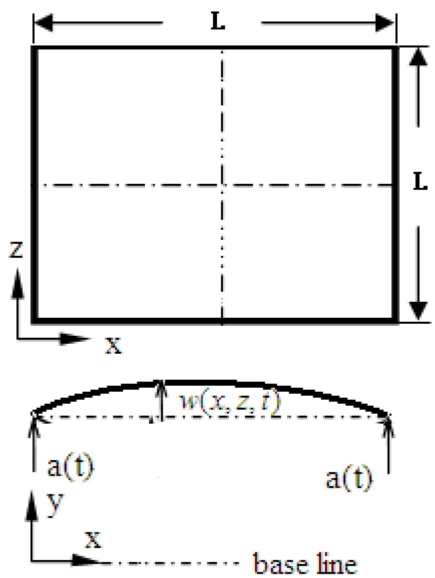

Fig.1. Simplified PCB model

Based on the simplified model, two failure mechanisms can be obtained: overstress fracture under high impact and fatigue fracture for multi-impact. For onboard devices, the launch environment take place only once, thus overstress fracture is concerned in this study.

The equations of motion under high-g impact are as follows (Suhir, 2002).

$$D\frac{\partial^4 w(x,z,t)}{\partial x^2} + \rho h \frac{\partial^2 w(x,z,t)}{\partial t^2} = 0 \tag{1}$$

Where $D = \frac{Eh^3}{12(1-\upsilon^2)}$ is the bending stiffness of PCB, ρ , E , h and υ are the density, elastic module, thickness and Posion rate of the substrate

The flexure deformation of PCB $w(x,z,t)$ can be defined as the linear superposition of model units, and the superposition equation is as follows:

$$w(x,z,t) = \sum_{i=1}^{\infty}\sum_{j=1}^{\infty} X_i(x) X_j(z) u_{ij}(t) \tag{2}$$

Where $u_{ij}(t)$ represents the displacement of micro-deformation unit corresponds to ij order mode shape function. The mode function $X_i(x)$ and $X_j(z)$ of PCB, the natural frequency ω_{ij} , and the displacement u of the board can be calculated respectively according to the following equation

$$X_i(x) = \sin(i\pi / a)x \tag{3}$$

$$X_j(z) = \sin(jz / b)z \tag{4}$$

$$\omega_{ij} = \pi^2 \gamma (i^2 + j^2) \big/ L^2 \tag{5}$$

$$\frac{u_{ij}}{a(t)}\omega_{ij}^2 = \frac{16}{ij\pi^2}\psi_{ij} \tag{6}$$

For most of the differential equations of motion, there is a large number of nonlinear terms, the general analytical solution cannot be obtained, and numerical solution is needed by computer analysis, FEM is commonly used (including explicit finite elements, implicit global model and transient dynamic response simulation such as LS-DYNA) and so on.

High impact test are normally conducted to understand the actual response and verify the analysis results. The actual acceleration response curves on different sites (impact table, base plane, and test vehicle) are acquired for further numerical analysis. (See next section)

3. FINITE ELEMENT MODEL AND DYNAMIC RESPONSE ANALYSIS

3.1 *The structure of MEMS*

In military applications, MEMS gyroscopes is mainly used for navigation guidance, attitude determination and stability, etc., the projectile usually stand with tremendous acceleration when launching, which would cause the solder joints fracture.

In this paper, a typical LCC package of MEMS gyroscopes in high impact is analyzed. The comb is installed inside the ceramic which is surface mounted on the PCB, brazing technology are used to cap sealing in vacuum condition. The package shown in Fig.2 is formed of the cover, ceramic, PCB and solder joints, the comb is ignored during the modeling for the mass and volume is much smaller than the package and the PCB.

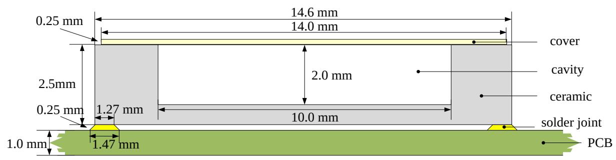

Fig.2. MEMS package structure and geometry

3.2 *FE modelling and analysis*

FE modeling is proven to be a very efficient tool for design analysis and optimization of IC and MEMS packaging, because of advantages of economic, saving time and being able to provide comprehensive information.

For the characters of high-g impact, the transient dynamic analysis is conduct using direct time-integration method. This can be further divided into implicit or explicit algorithm. While the contact duration may be extremely short and intense or when the impacting bodies if interest are excited into very high frequencies response, such as the cases of drop impact on rigid surface. For all of these reasons, the explicit algorithm is adopted in this analysis.

Finite element model was established according to the MEMS gyroscope. The structure is mainly combined of the cover, ceramic, PCB and solder joints. Solid element (SOLID 164) was used to model all components including the PCB in this model. Due to symmetrical of the package, a quarter of the model was established in order to simplify modeling and save compute time, finite element model of LCC package is shown in Fig.3.

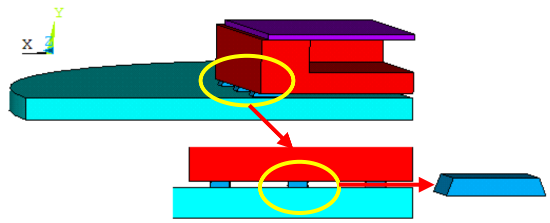

Fig.3. Finite element model of MEMS package

The material properties of MEMS packaging used in the FEM are shown in Table 1. The bilinear kinematic hardening model is chosen for the solder joints to be closer to the engineering practice, and linear material model is used for the rest component of the package. Generally, the damping of the system is often set between 0.01-0.25, where a fixed value of 0.03 is recommended here (Zhao et.al., 2004).

Table 1 Material properties used in FEM

Structure	*Materials*	*Density (kg/m^3)*	*Young's modulus (GPa)*	*Poisson's*	*Yield strength (MPa)*	*Tangent modulus (GPa)*
PCB	FR4	1900	22	0.3	450	—
Cover	alloy	8460	138	0.31	380	—
Body	ceramal	3920	344	0.22	580	—
Solder joint	Sn3.5Ag	7400	52.7	0.4	22.5	3.09

The hexahedral element is used to mesh the FEM in this paper for the low stiffness and high accuracy compared to the tetrahedron element. The mesh of the solder joints are refined to get a precise result of the stress in solder joints. The model is consisting of 10,772 nodes and 7,371 units.

The input-G method developed by Tee (Tee *et.al.*, 2004) is used in this dynamic analysis. This method is more accurate and much faster, and bypasses many technical difficulties in conventional dynamic model such as adjusting the parameters of contact surfaces, defining contact type, etc. In this way, only the package itself needs to be modeled. The impact acceleration pulsed which is measured and built from the actual test from the missile (see Fig.4.), are imposed on the supports of the board-level test vehicle as load condition. A very detailed finite element model of the board and package was constructed and simulated by LS-DYNA (ANAYS).

Fig.4. shows the impact pulse according to the actual measurement which features as an impact acceleration pulse with a duration of 0.01s, the initial value of $1.44\times10^4 m/s^2$ and peak acceleration of 12600g at 0.0026s, at 0.01 seconds, the acceleration reduced to $4.71\times10^4 m/s^2$, after 0.01s, the load was removed.

In ideal conditions, MEMS packaging only produced a straight up displacement (y direction) and the displacement (x, z) of PCB is confined in the whole process during the impact. As the result of a quarter models, symmetry boundary constraints should be applied to xy and yz surface on the inner side of the model.

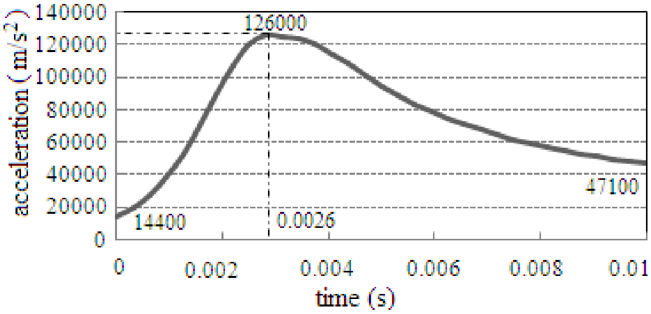

Fig.4. Acceleration curve

Explicit nonlinear algorithm, a more suitable method, was used for dynamic response analysis. Contact surface cannot penetrate each other by the outer surface, so automatic single surface contact was set for global contact. Hourglass is a model in theory, not exist in the actual process, it is a mathematically stable but physically impossible state. Solutions will be useless because of hourglass, and if the total hourglass of the model can be greater than 10%, the result is generally a fault result. The hourglass was set 10% of the total internal energy in this paper. The results file output interval was 100, and time-history output Interval was 100, the solution termination time was 0.03s.

3.3 *Results and Discussion*

Fig.5 shows the typical simulation analysis results for the contours of von-mises stress of MEMS. The response of MEMS package under high-g impact is a dynamic process, the maximum effective stress occurs in the outermost corner and reached to 27.17MPa, more than the yield strength of solder joint (22.5MPa), which make the solder joint produce plastic deformation and maybe fracture during the impact; The maximum plastic strain has reached 2.7×10^{-3}. Fig.6 shows Time-History curve of solder joint at the maximum stress point. The solder joints experience tensile stress first, and then stress changes to attenuating. The effective stress reached its peak at 0.0027s, and after about 0.0105s, the stress tends to stabilize.

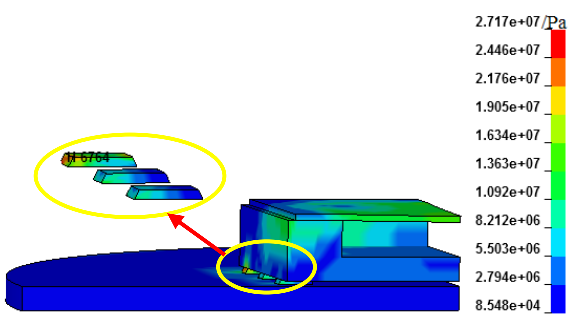

Fig.5. The stress distrubuting of the finite element simulation

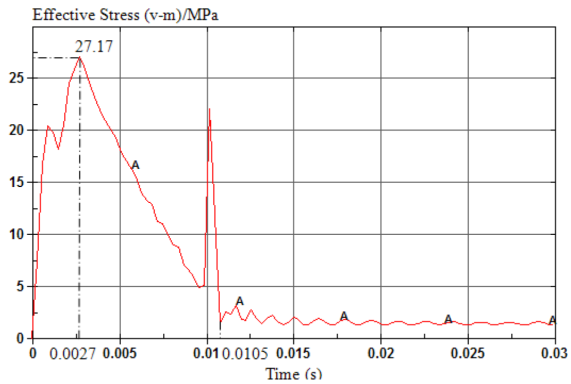

Fig.6. Effective stress Time-Histroy of maximum stress point

By comparing the deflection-time curve of the PCB and the stress-time curve of the solder joints, it finds that the maximum stress in solder joint reaches its peak at the moment of the deflection of the PCB reaches its maximum, and the effective stress has a slight fluctuation along with the acceleration curve during the impact. It suggests that peeling stress in solder joints is mainly caused by the deflection of the PCB board during impact.

In addition, the deformation of the alloy cover is large during the impact; Fig.7 shows the cover had bending phenomenon to the cavity. At the same time, difference between the maximum and minimum displacement of the unit is the maximum deformation, the maximum deformation of the cover is 0.08mm, which is 32% of the cover thickness (0.25mm), and it is 4% of the cavity depth MEMS devices (2mm), if the deformation of alloy cover is too large, it will squeeze the internal structure in LCC package, then failure occurs. Therefore, the proposed design of the LCC package should reserve larger space for the upper part to avoid excessive deformation of the cover. In addition, by using potting processing, the squeeze effect would be more serious.

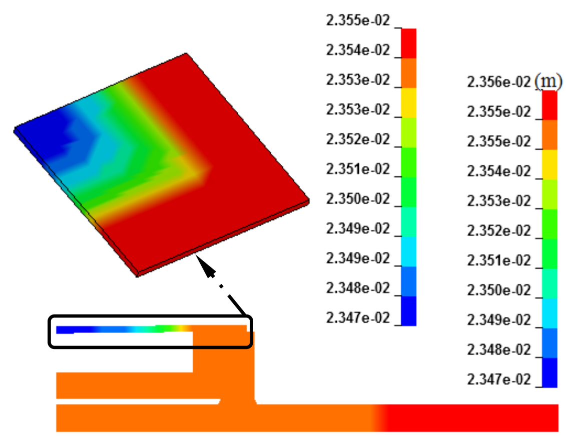

Fig.7. The deformation distrubuting of the cover

Meanwhile, the edge of PCB occurred to bend up, as shown in Fig.8. And at the moment of maximum stress occurred, the deformation of PCB reached a maximum of 0.01mm, which is 1% of its own thickness (1mm), and it is 5% of solder joint height (0.2mm), thus, the squeeze will extrusion the solder joint which is the main reason to produce internal stress of the solder joint.

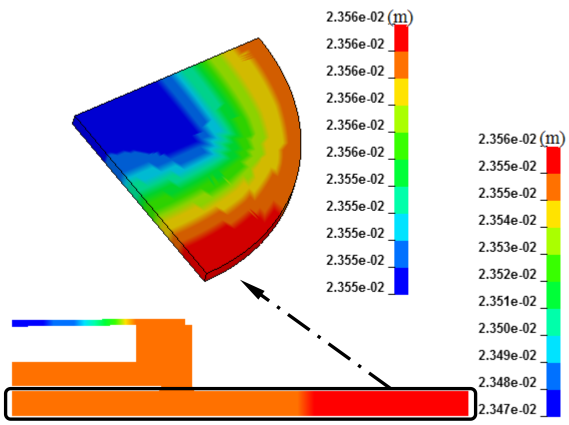

Fig.8. The deformation distributing of the PCB

4. CONCLUSIONS

This paper conducts a research on the dynamic response of MEMS gyroscope board-level package reliability under high-G impact. Theoretical analysis and mathematical model for failure mechanism of MEMS package (Leadless Chip Carrier, LCC) under high-G impact are established and analyzed. Analytical solutions that provide physical insights to the dynamics of PCB and the interconnection stresses have been presented.

Under high-g impact, solder joint is the key weakness that influence MEMS package, which will be fractured and failure easily. The response of MEMS package is a dynamic process. Moreover, the maximum effective stress in the structure occurs at the outer corner in the outermost solder point, and the alloy cover and PCB have a greater deformation.

The room between the cover and the device inside MEMS should be carefully designed, and the distant must be longer than 10% of the cavity depth.

Considering the failure mechanism of solder joint that the flexure deformation of the PCB produces stress inner solder joint, we recommend minimizing the length and width of the PCB or increasing the thickness of the PCB or increasing the stiffness of the PCB so as to decrease the stresses in the solder joint.

ACKNOWLEDGEMENT

This study was conducted under the contact of NanJing University of Science and Technology. The author also would like to thank the member of City PHM Team and the reviewers of this paper for their valuable suggestions.

REFERENCES

Lou Xia, Shi Jinjie, Zhang Wei, et al. Study on the Packaging Technology for a High-G MEMS Accelerometer [C]. *Proceedings of Electronic Packaging Technology Conference, Singapore*: IEEE, 2005, 103-106

Vinod Chakka, Mohamed B Trabia. Modeling and Reduction of Shocks on Electronic Components within a Projectile [J]. *International Journal of Impact Engineering*, 2008,35(11) 1326-1338

Jiang Qiyu, MaoHu Du, Le Luo, et al. Simulation of the Potting Effect on the High-G MEMS Accelerometer [J]. *Journal of Electronic Materials*,2004,33(8):893-898

Wei Lingyun, Mei Zhao, Qiang Guo et al. SMT Solder Joint's Shape and Location Optimization using Modified Genetic Algorithm in the Dynamic Loadings [C]. *Proceedings of the International Conference on the Business of Electronic Product Reliability and Liability*, 2004, 169-173

Thomas F. Marinis, Joseph W. Soucy and James G. Lawrence. Vacuum Sealed MEMS Package with an Optical Window [C]. *Proceedings of Electronic Components and Technology Conference*, 2008, 804-810

Tee T Y, Luan Jing-en. Advanced Experimental and Simulation Techniques for Analysis of Dynamic Responses during Drop Impact [C]. *Proceedings of the 5th Electric Component and Technology Conference, Las Vegas*: IEEE, 2004,1088-1094

JEDEC Standard JESD22-B111. Board Level Drop Test Method of Components for Handheld Electronic Products[S], 2003

Younis, Daniel Jordy, et al. Computationally Efficient Approaches to Characterize the Dynamic Response of Microstructures under Mechanical Shock [J]. *Journal of Microelec-tromechanical systems*, 2007:16(3), 628-638

Sirkar V T, Senturia S D. The Reliability of Microelectromechanical System (MEMS) in Shock Environments [J]. *Journal of Microelectromechanical systems*, 2002, 11(3):206-214

Suhir E, Burke R. Analysis and Optimization of the Dynamic Response of a Rectangular Plate to a Shock Load Acting on its Support Contour with Application to Portable Electronic Products [J]. *Advanced Packaging*, 2000, 122(1):3-5

Yu Q, Kikuchi H, Ikeda S Y, et al. Dynamic behavior of electronics package and impact reliability of BGA solder joints [J]. *Structure and Materials*, 2003, 12:55-64

Tee T Y, PEK Eric, et al. Novel Numerical and Experimental Analysis of Dynamic Response under Board Level Drop Test [C]. *Proceedings of 5th International Conference on Thermal and Mechanical Simulation and Experiments in Microelectronics and Microsystems, Brussels*: IEEE, 2004, 1131-1124

Tao Jianlei, Qu Xin, Wang Jiaji. Simulation of Average Strain Energy Density (SED) in the BGA Soldering during the Drop Test [C]. *Proceedings of 7th International Conference on Electronics Packaging Technology*, Shanghai: 2006,1-6

Wong E H. Drop impact test - mechanics & physics of failure

Tee T Y, Ng H S, Zhong Z P, et al. Design for Enhance Solder Joint Reliability of Integrated Passives Device under Board Level Drop Test and Thermal Cycling Test [C]. *Proceedings of 5th EPTC Conference*. Singapore: IEEE, 2003,210-216

E Suhir. Could Shock Tests Adequately Mimic Drop Test Conditions? [C] *Proceedings of Electronic Components and Technology Conference. Singapore*: IEEE, 2002,563-573

Zhao Mei, Zhou Haitang, et al. Mechanical vibration and noise [M]. *Beijing: Science and Technology Press*, 2004(In Chinese)

Jiuzheng Cui is currently a student of School of Reliability and Systems Engineering at Beihang University in Beijing, China. He received his B.E. degree in detective guidance and control from Northwest Polytechnic University. His current research interests include reliability of electronics, physics of failure, prognostics and health management, failure analysis of electronics, reliability engineering, and integrated design of product reliability and performance.

Dr. Bo Sun is a reliability engineer and a member of the faculty of School of Reliability and Systems Engineering at Beihang University in Beijing, China. He received his Ph.D. degree in reliability engineering and systems engineering from Beihang University and a B.S. degree in mechanical engineering from the Beijing Institute of Mechanical Industry. His current research interests include reliability of electronics, physics of failure, prognostics and health management, failure analysis of electronics, reliability engineering, and integrated design of product reliability and performance.

Dr.Qiang Feng is a reliability engineer and a member of the faculty of School of Reliability and Systems Engineering at Beihang University in Beijing, China. He received his Ph.D . degree in systems engineering and a B.S. degree in mechan ical engineering from the Beihang University. His current re search interests include reliability engineering, systems engi neering,physics of failure, and synthetical design of product reliability maintainability supportability and performance.

Prof. ShengKui Zeng is currently the Vice Director of the School of Reliability and Systems Engineering at Beihang University in Beijing, China. He has over 15 years of research and teaching experience in reliability engineering and systems engineering. He was a visiting researcher with the Center for Advanced Life Cycle Engineering Electronic Products and Systems Consortium at the University of Maryland in 2005. He is the team leader of the KW-ARMS© reliability engineering software platform, the co-author of three books, and a recipient of three Chinese ministry-level professional awards. His recent research interests include prognostics and health management, integrated design of reliability and performance, and reliability-based multidisciplinary design optimization.