Stress analysis of automotive IGBT module under vibration load

1st Yanzhong Tian
Institute of Electronics Packaging Technology and Reliability
Beijing Key Laboratory of Advanced Manufacturing Technology
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
tianyz@emails.bjut.edu.cn

2nd Tong An*
Institute of Electronics Packaging Technology and Reliability
Beijing Key Laboratory of Advanced Manufacturing Technology
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
antong@bjut.edu.cn

3rd Fei Qin
Institute of Electronics Packaging Technology and Reliability
Beijing Key Laboratory of Advanced Manufacturing Technology
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
qfei@bjut.edu.cn

4th Yanpeng Gong
Institute of Electronics Packaging Technology and Reliability
Beijing Key Laboratory of Advanced Manufacturing Technology
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
yanpeng.gong@bjut.edu.cn

5th Yanwei Dai
Institute of Electronics Packaging Technology and Reliability
Beijing Key Laboratory of Advanced Manufacturing Technology
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
ywdai@bjut.edu.cn

6th Yanpeng Chen
Institute of Electronics Packaging Technology and Reliability
Beijing Key Laboratory of Advanced Manufacturing Technology
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
pccn@vip.163.com

2022 23rd International Conference on Electronic Packaging Technology (ICEPT) | 978-1-6654-9905-7/22/$31.00 ©2022 IEEE | DOI: 10.1109/ICEPT56209.2022.9873277

***Abstract*—With the development of new energy vehicles, its control component (automotive IGBT module) has met new opportunities and challenges. Automotive IGBT modules are in severe service conditions and often suffer from vibration environment. Therefore, the vibration analysis of automotive IGBT module has important practical significance. In this paper, finite element software is used to study the stress distribution of automotive IGBT module under two vibration modes: simple harmonic vibration and random vibration, and a comparison is made. By observing the stress nephogram of simple harmonic vibration and random vibration , it can be found that the stress in the middle of the module and the root of the bonding wire is larger in both vibration modes.**

Keywords—Insulated gate bipolar transistor (IGBT), Reliability, Vibration, stress analysis, Finite-element method (FEM)

I. Introduction

In recent years, with the vigorous development of the automotive industry, the automotive IGBT industry has ushered in new opportunities as well as new challenges. Compared with the general IGBT modules, the working condition of the automotive IGBT modules is more complex and requires higher reliability. The service environment of automotive IGBT module is very bad, and it often has to withstand severe temperature and mechanical load (vibration and impact). The investigation results show that the failure rate of electronic equipment caused by working environment factors is as high as about 50% [1], among which the vibration factor accounts for about 27% of the environmental factors that cause the failure of electronic equipment [2].

In the field of aircraft and automobile, many products have to experience vibration environment during service. In many vibration environments, simple harmonic vibration is the simplest and random vibration is the most common [3]. Simple harmonic vibration is the most basic vibration. Any periodic vibration can be synthesized by several simple harmonic vibrations. Random vibration is the most complex vibration, and the analysis of random vibration needs the help of probability. Random vibration analysis, also known as power spectral density analysis, is a spectral analysis technology based on probability statistics, which transforms statistical samples of time history into power spectral density (PSD) [4].

In this paper, the whole stress analysis of automotive IGBT module under vibration load is studied by numerical simulation, and the comparison is made. Modal analysis is the first step for vibration analysis, and then the simple harmonic vibration is swept by the first-order modal natural frequency. PSD curve is selected for random vibration according to the vehicle road environment manual, and the maximum acceleration is 20g. It can be seen from the overall vibration stress nephogram that no matter it is simple harmonic vibration or random vibration, the maximum stress is mainly generated at the fixed position of the copper substrate and the ceramic layer, but these positions are protected by packaging shell and silica gel, which is not easy to fail. In addition, the stress concentration mainly occurs at the junction of bonding line and aluminum metallization layer.

II. Vibration Theory

Simple harmonic vibration is the simplest vibration, which only occurs in laboratory conditions. Random vibration is the most complex vibration, which is the most common in nature. Next, Taking the single-degree-of-freedom system as an example, the simple harmonic vibration and random vibration theories are introduced respectively. The schematic diagram of the single degree of freedom system is shown, including springs, mass blocks and dampers.

This research was supported by the National Natural Science Foundation of China (NSFC) No. 11872078

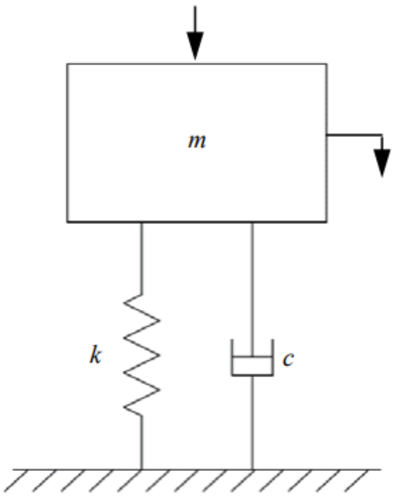


Fig.1. The schematic diagram of single degree of freedom system

A. Simple Harmonic Vibration

Under the action of simple harmonic excitation ($F\sin\omega t$), the motion equation [5] of the single degree of freedom system is

$$m\ddot{x}+c\dot{x}+kx=F\sin\omega t \tag{1}$$

where m is mass, c is damping coefficient and k is stiffness coefficient.

The solution of this non-homogeneous differential equation is

$$x = e^{-\xi\omega_n t}(A\cos\omega_d t + B\sin\omega_d t) + \frac{F\sin(\omega t-\varphi)}{\sqrt{(k-m\omega^2)^2+c^2\omega^2}} \tag{2}$$

where ξ is damping ratio, ω_n is natural frequency, ω_d is damping resonance frequency, A and B are constants, φ is phase angle.

B. Random Vibration

Under random excitation $f(t)$, the response of the system calculated by convolution [5] is

$$x(t) = f(t)*h(t) = \int_{-\infty}^{+\infty} f(\tau)h(t-\tau)d\tau \tag{3}$$

where $h(t)$ is system unit impulse response.

After a series of complex calculations, and the introduction of power spectral density $S_f(\omega)$, we can get

$$\bar{x}^2 = E[x^2(t)] = \frac{1}{2\pi}\int_{-\infty}^{+\infty} |H(\omega)|^2 S_f(\omega)d\omega \tag{4}$$

where E is the root mean square and $H(\omega)$ is the Fourier transform.

III. Finite Element Analysis

A. Module Introduction

The research sample is an automotive IGBT module of BOG450H12E2AA, with a rated current of 450 A and a rated voltage of 1200 V. The actual appearance structure is shown in Fig. 2. This module is composed of upper and lower half bridge arms, each half bridge arm contains 3 IGBT chips and 3 FWD diode chips. The upper and lower half bridge arms are connected in series to form a full bridge. Each IGBT chip is connected in parallel with a diode chip to realize freewheeling. In addition, the external terminals used to connect external electrical signals form the internal circuit of the whole module.

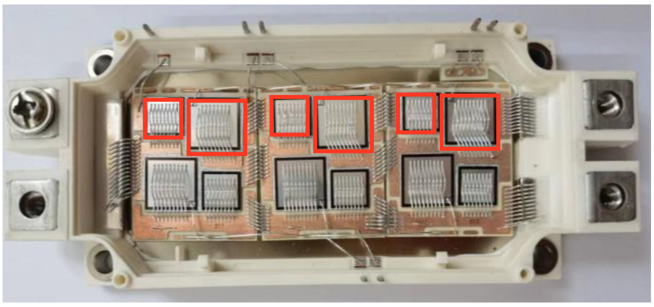


Fig. 2. The actual appearance structure

B. Establishment of Finite Element Model

As mentioned above, the main structures of automotive IGBT modules include al bonding wire, Al metallization layer, IGBT chip, diode FWD chip, chip solder layer, DBC substrate, substrate solder layer, Cu substrate and packaging shell. According to the actual size of automotive IGBT module, a three-dimensional model is established in the finite element analysis software, as shown in Fig. 3. In order to simplify the calculation, reduce the complexity of the model and save the calculation time, the simplification is made when establishing the three-dimensional finite element model. After the model is established, assign material parameters to each layer, as shown in Table I.

TABLE I MATERIAL PROPERTIES

Materials	Elastic modulus (GPa)	poisson ratio	Density (kg/m³)
Aluminum	70	0.33	2.70×10^3
Silicon (IGBT)	112	0.22	2.33×10^3
Solder (SAC305)	10.6	0.35	7.30×10^3
Ceramic (Al_2O_3)	112	0.22	3.96×10^3
Copper	10.6	0.34	8.92×10^3

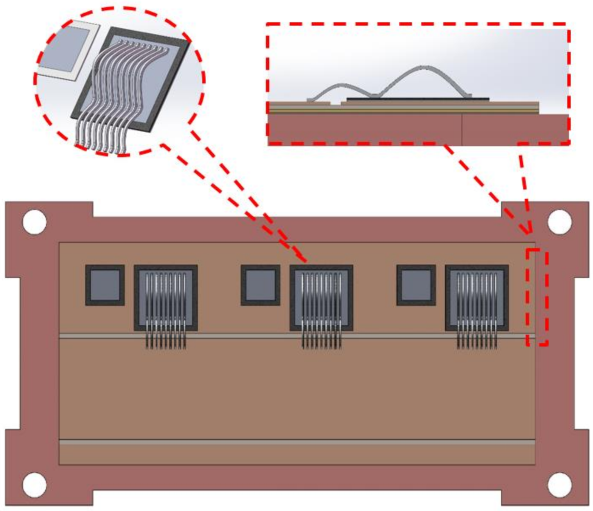


Fig. 3. 3D finite element model diagram

One of the most important steps in finite element analysis is mesh generation. The hexahedral element (C3D8R) is used as the mesh. After verifying the convergence of the mesh, the three-dimensional model is divided into 578642 elements and 665535 nodes.

C. Loads and Boundary Conditions

In the vibration analysis, in order to enhance the anti-vibration characteristics and prevent resonance, a fixing method consistent with the actual working state, i.e. four corner bolt fixing, is selected. Under the condition of simple harmonic vibration, the load is a swept sinusoidal vibration with a frequency of 900-1000 Hz. Under random vibration conditions, the PSD curve is shown in Fig. 4, and the maximum acceleration is 20g.

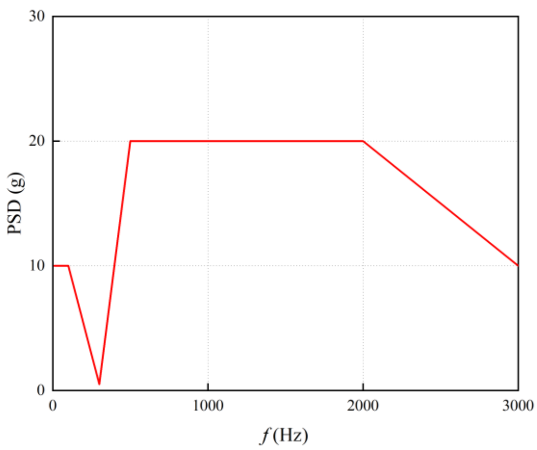


Fig. 4. The PSD curve

According to the harmonic vibration conditions, the inner wall of the bolt hole in the shell of the automotive IGBT module is fully constrained and loaded by the method of foundation excitation. By loading a sinusoidal vibration load with an acceleration amplitude of 20g and a sweep frequency range of 900-1000 Hz on the boundary of the constraint, the vibration direction is perpendicular to the module surface. Under random vibration conditions, the fixed constraints are the same, and the acceleration PSD curve is applied to the module for random vibration response analysis.

IV. Finite Element Result Analysis

A. Modal Analysis

The objective of modal analysis is to provide basis for system vibration characteristic analysis, fault diagnosis and prediction, dynamic characteristic optimization, etc. through the modal parameters of the system. With the continuous development of electronic equipment, modal analysis has become more and more critical, and has become a necessary step to solve the vibration problem. Supplemented by computer technology and experiments, it is a key link in product design and detection.

The first five modes of the model are simulated by using the finite element software for modal analysis, and their natural frequencies are shown in Table II. The model size of automotive IGBT module is small, the material stiffness is large, and the overall mass is small. Therefore, it can be judged that the modal frequency of the model is large. The research shows that in the structural vibration of the model, the energy of higher-order modes accounts for a relatively low proportion and has little impact on the vibration of the whole structure, so the vibration load with low vibration frequency is the key. Among them, the most important first-order natural frequency is 957.8 Hz.

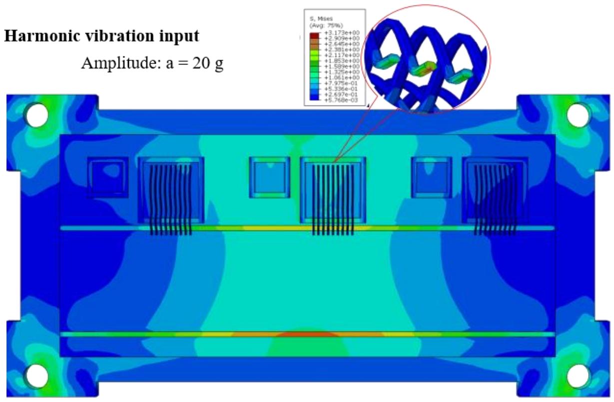
TABLE II MODAL ANALYSIS

Modal/rank	Frequency/Hz
1	957.8
2	1830.9
3	2312.1
4	3811.7
5	4099.2

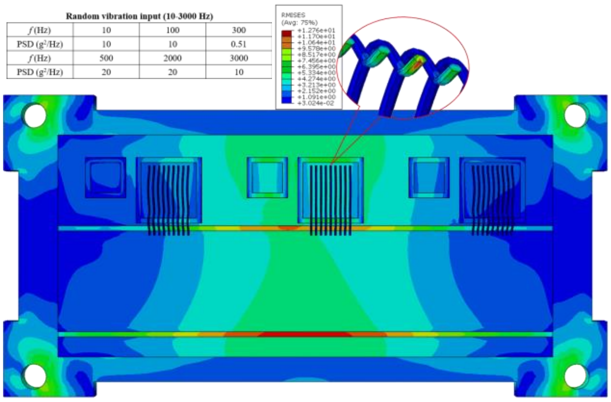
B. Vibration Response Analysis

The modal analysis results and the two vibration conditions are combined to carry out simple harmonic vibration and random analysis respectively, and the equivalent stress of automotive IGBT module under simple harmonic vibration and random vibration conditions is obtained, as shown in Fig. 5. It can be seen from the overall stress distribution that the equivalent stress of the module is mainly concentrated in the fixed hole of the copper substrate and the ceramic layer in the DBC, whether it is simple harmonic vibration or random vibration. However, due to the protection of the packaging shell and silica gel in the fixed hole of the copper substrate and the ceramic layer, the influence of the stress concentration will not produce fatigue failure. In addition, the other stress is larger at the root of the bonding line. At the junction of bonding wire and aluminum metallization layer, the stress concentration at the bonding wire bonding is most likely to cause the bonding failure of the module. The bonding line material is aluminum, and the bonding line is thin, the structural strength is low, which makes it a weak part of the whole module structure.

It can be seen from the stress nephogram that under the condition of simple harmonic vibration, the maximum MISES stress at the root of the bond line is 3.17 MPa, and under the condition of random vibration, the maximum MISES stress at the root of the bond line is 12.76 MPa. By comparison, it can be found that the stress generated at the root of the bond line under random vibration is greater, and it is easier to generate stress concentration. Therefore, the probability of failure of the automotive IGBT module under random vibration is greater.



(a)



(b)

Fig. 5. Stress distribution (a) under harmonic vibration load, (b) under random vibration load.

V. CONCLUSION

In this paper, the reliability of automotive IGBT module under vibration conditions is studied. The vibration response analysis of automotive IGBT module under simple harmonic vibration and random vibration is carried out by numerical simulation, and the following conclusions are obtained.

(1) Through the stress nephogram, it can be found that the stress distribution of IGBT module caused by vibration is basically the same whether it is simple harmonic vibration or random vibration.

(2) The stress is the largest in ceramics layer, but ceramics have extraordinary stability. For the bonding wire we are concerned about, the maximum stress appears at the root of the bonding wire, so the failure of the bonding wire should be paid more attention.

ACKNOWLEDGMENT

This research was supported by the National Natural Science Foundation of China (NSFC) No. 11872078.

REFERENCES

[1] Hienonen R, Karjalainen M, and Lankiene R. “Verification of the Thermal Design of Electronic Equipment,” Espoo: Technical Research Centre of Finland. 1997, pp. 13-86.

[2] J. Jones and J. Hayes, “Investigation of the occurrence of: no-faults-found in electronic equipment,” IEEE Transactions on Reliability, vol. 50, no. 3, pp. 289-292, September 2001.

[3] Yu D, AL-Yafawi A, Park S, and Chung S. “Finite element based fatigue life prediction for electronic components under random vibration loading,” Proceedings-Electronic Components and Technology Conference. Las Vegas, pp. 188-193, June 2010.

[4] N. G. Bingel III; C. C. Bleakley, A. L. Clapp, J. H. Dagenhart, M. A. Konz, L. E. Gaunt, M. B. Gunter and M. F. Jurgemeyer, “2017 NESC(R) Handbook, Eighth Edition,” in National Electrical Safety Code (NESC) Handbook, Eighth Edition, pp.1-428, August 2016.

[5] Ronald Huston, and C Liu, “Principles of Vibration Analysis with Applications in Automotive Engineering,” Principles of Vibration Analysis with Applications in Automotive Engineering , SAE, 2011.