Indian Journal of Fundamental and Applied Life Sciences ISSN: 2231– 6345 (Online)
An Open Access, Online International Journal Available at www.cibtech.org/sp.ed/jls/2014/04/jls.htm
2014 Vol. 4 (S4), pp. 381-388/Zandipour and Jalali
Research Article

MODELING AND VIBRATION ANALYSIS OF ELECTRONIC EQUIPMENT IN AEROSPACE STRUCTURES

***Hamid Reza Zandipour and Hasan Jalali**
Iran University of Science and Technology Narmak, Tehran, 16846, IRAN
**Author for Correspondence*

ABSTRACT
Aerospace structures usually pass through different environments in their launch and motion course. While passing through these environments different loads and boundary conditions are ap-plied on them. On one hand, the existence of sensitive cargos in these structures have caused that recognition of dynamic loads along the course and control and separation of undesired vibration and the decrease of transmitted vibration level to this sections seem a necessary issue. Therefore, in this article dynamic parameters of electronic boards have been recognized and analyzed. For this pur-pose electronic boards with printed circuit board (PCB), which are used in different industries in-cluding aerospace structures, have been modeled using finite element method. After modeling available structure and its joints in the finite element model, the finite element model was updated using experimental results from modal testing and finally it has been used of a corrected model for calculating dynamic stresses in joints.
First, by using Calfem software, the finite element model of simple plate was made without in-stalling Thin Quad Flat Pack (TQFP) piece. Then, plate model with one TQFP piece was made. For modeling the joint between the piece and plate, a combination of linear and torsional spring was used. In addition, the real sample of considered board was prepared and by using of modal testing its dynamic parameters including natural frequencies and damping coefficient for simple board and board including TQFP piece were identified. Then the finite element model of board corrected. For this purpose, the finite element model by minimizing the error between natural frequencies resulted from the test and finite element model of board by Eigen value sensitivity approach was corrected. To model damping, proportional damping model for finite element model of simple board was con-sidered and by minimizing the difference between damping ratios resulted from the test and finite element model, coefficients of proportional damping model obtained. Finally, finite element model of board including TQFP piece was corrected by using of natural frequencies resulted from the test. By having an updated finite element model, dynamic stresses in solder joints under specified excita-tion condition has been calculated.

***Keywords**: Printed Circuit Boards; Finite Element Model; Solder Joint; Updating*

INTRODUCTION
The packages of printed circuit board is one most important components of these electronic packages used in the aerospace structure that these boards include a PCB along with other electronic components mounted on it such as TQFP and IC and other attached pieces. In this research as a sample, a model of a simple PCB along as well as a TQFP piece mounted on it has been considered. The general aim of formulating this article is creating an exact finite element model of PCB includ-ing a TQFP piece in order to obtain dynamic stresses in the solder joints of TQFP piece under con-ditions similar to real conditions. Doing this due to the direct measurement of stress by using of strain gauge installation is impossible because of the smallness of TQFP joints size, has very high importance. In the first studies of electronic equipment vibration analysis, the researchers seek to make an appropriate mathematical model for simple and complex electronic pieces. The first book that was published in this filed was Electronic Equipment Vibration Analysis written by Stein-berg (Steinberg, 1988). The first publication of this book was in 1973 and its second edition published in 1988. This book is the base of electronic equipment modeling such as multi degree freedom of concentrated mass, beams, frames, flat board and so on. The above-mentioned

modeling methods are applicable for designer engineers in connection with electronic boards. For electronic packages which every day become more complex, making an appropriate model for their dynamic and vibration analysis has been paid attention by engineers. Wong (Tin-Lup Wong et al, 1990) performed the first researches in this relation. He performed researches about the respond of surface mounted technology (SMT) pieces on the printed circuit board under bending and torsional loading. The main important subject in these studies was determining the connection between input load in the solder joint and obtained force from drop test. In these studies, it was used of finite element method for determining the PCB displacement. Lau and Rice (John H Lau and Rice W. Donald, 1985) reviewed different methods for analyzing the fatigue of solder joints between SMT pieces and PCB boards and in connection with physical and mechanical specification of solder joint collected useful information. However, the existed hypotheses did not change but they did numerous researches about this subject. Ju-Sandor and Plesha (Ju, Sandor & Plesha 1996) proposed two finite element method for refraction analysis and evaluating fatigue age of solder joints. The main subject of these studies was doing research on reliability of solder joints under thermal loads. Lau (Lau et al, 1989) also did very researches on reliability of solder joints under shock and vibration loads resulted from thermal cycle change. In these researches, it was obvious that interred vibrations on PCB vertically have more impact on the reliability of solder joints in proportion to board vibrations. In addition, it was determined that refraction time is under two factors of natural frequencies of PCB and excitation frequencies. Pittaresi(Pitarresi et al, 1991) in a research that did in 1991 provided a method for identifying mechanical properties of piece mounted on a PCB as well as its dynamic finite element analysis that can achieve fine and acceptable results from natural frequencies and electronic components modes form by simplifying complex models. He compared the experiment results with finite element analysis results used Modal Assurance Criterion (MAC) method. Xin Jun Shen (Zhang, Ding & Sheng, 2008) and his colleagues through doing modal test and finite element analysis and updating to identify dynamic parameters of PCB by mounting electronic pieces and by exciting and measuring special points in these structures have obtained fine results. Jing-en Luan [8] and his colleagues by using of drop test method wanted to find dynamic stresses in the solder joints and to find fatigue age of these joints have done wide researches. Due to the sensitivity of such pieces in performing aerospace missions, to provide better methods and models in this connections research is continued. In this article, it has been used of methods and articles of authority and they have been referred at the references completely.

Finite Element Modeling

As was mentioned before, a standard sample of PCB simple board applied in aerospace structures with 0.14× 0.21 meter and 1.5 mm thickness has been selected. Primary mechanical properties considered for this board has been offered in table (1). It is necessary to mention that in the next sections by using of updating method exact properties of materials is determined in this page. The kind of this board is FR4 material, which orthotropic material.

Table 1: Mechanical properties of PCB simple board (FR4 kind)

Density ρ(Kg/m3)	υxy=υyx	(GPa) Gxy	(GPa)Eyy	(GPa) Exx
1971	0.12	5.5	25	30

The model of this page was created in Calfem (Luan at al, 2007) software by 660 elements of four-node plate as well as 713 nodes. The total number of freedom degree of this model is 2139. The number of used elements in this model in the length direction is 30 elements and in the width direction is 22 element that finally 660 elements is produced for finite element model of simple board of PCB. The dimension of stiffness and mass matrix for this 2139 degree of freedom model is 2139 × 2139. By making this model stiffness and mass matrix of finite element model of PCB simple board is available for dynamic analysis. Then, a proper model of finite element for a board that a TQFP piece has been mounted on it also is produced. This section is done in two stages. In the first stage TQFP piece is modeled and then a proper model for the connection between the board and piece is considered and with such supposition the connection between the board and piece is modeled. The dimensions of considered TQFP piece according to the standard are 0.027 m × 0.027 m with 3 mm thickness .Mechanical properties of piece in the primary supposition was considered equal to 0.1 of aluminium mechanical properties. In order to that the nodes of piece and board coincide, the piece is divided to 9 equal elements. In this condition the total number of model elements along with PCB simple board that prepared beforehand reaches 669 elements of 4-node board and the total number of model's nodes will be 729. The degrees of freedom whole model are 2178. Modeling TQFP piece and PCB board has been done but for modeling the connection, it is necessary to consider proper model. There are different methods for modeling this connection. Among these methods using linear and torsional spring model for this connection of TQFP piece to PCB board is more proper. Therefore in modeling a board with a TQFP piece, modeling the solder joint between TQFP piece and PCB board with 12 linear spring in Z direction and 12 torsional spring in the two directions of X and Y is done.

Testing and Updating

Modal Test

Firstly, real sample of PCB simple board in 0.14 m × 0.21 m dimensions and 1.5 mm thickness and a board including one TQFP piece was prepared. The dimension of TQFP piece is 0.027 × 0.027 m and its thickness 3 mm. in order to create free boundary conditions the structure with high flexibility threads has been hanged. A hammer excites the structure and the response of structure measurement is done by a piezoelectric accelerometer. The accelerometer has attached to the PCB board by wax, therefore if it is necessary the measurement point changes easily. So, all modes of system are recognized. To excite the system, it has been used of hammer (Ewins, 2000). Through doing modal test on the structure frequencies response functions (FRFs) are measured. At first, the test related to simple board is done. In the next test, the board constructed sample with a TQFP piece with the same dimensions of finite element model was tested by modal.

Updating Finite element model of PCB Simple Board with Modal Test Results

To update finite element model of simple board it has been used of Eigen values sensitivity approach. As was mentioned beforehand, the difference between natural frequencies of primary finite element model and natural differences resulted from modal test because of the uncertainty of material properties used in the finite element model. From frequency response functions diagrams of PCB simple board, the first seven frequencies will be used for updating. In table (2), natural frequencies of PCB resulted from modal test along with results obtained from reanalyzing finite element model with updated results are provided. In this table can observe the error percent, too. As it is seen in table (2), in the first four mode of model, the error percent is little and in the first seven mode of model, the error percent is acceptable.

Table 2: The comparison of natural frequencies of finite element model of PCB simple board and test

Mode number	1	2	3	4	5	6	7
Updated	90.564	123.742	221.839	284.982	329.614	350.575	437.958
Experimental	92	124	224	290	328	364	434
Error (%)	0.3856	0.1864	0.1174	-0.4276	-1.4700	2.7620	-1.4322

Updating Finite element Model of Board with a TQFP Piece

In table (3), natural frequencies of board with a TQFP piece resulted from modal test along with obtained results from reanalyzing finite element model with updated results are offered. In this table can observe the error percent, too. As it is seen in the table, the error of first, third, fourth, fifth, sixth, and seventh mode is similar and acceptable. However, the error of second mode in relation to the six other modes is further. The reason of this high error is that in this mode the TQFP piece is located on a node, therefore the changes of designing parameters do not have high impact on the natural frequency.

Table 3: Comparing natural frequencies of finite element model of board with a TQFP piece and test

Mode number	1	2	3	4	5	6	7
Updated	93.279	125.904	226.193	287.962	335.004	354.906	406.277
Experimental	98	136	232	288	334	368	434
Error (%)	3.4101	11.1466	2.5961	-2.9061	1.3080	3.2653	6.8325

The obtained finite element model until this stage is able to stimulate natural frequencies resulted from the test. Because this model will be used for dynamic analysis of board and have important role in damping dynamic analysis, damping model of this model also should be corrected. This will be done in the next section.

Obtaining Damping Model of PCB Simple Board Model

By comparing frequency response functions diagrams of two simple boards model and board with a TQFP piece in the previous section it was recognized that TQFP piece do not make any change in the simple board damping and only it is effective in changing natural frequencies of simple board model. For this reason can obtain a proper damping model for PCB simple board and use it for board model with a TQFP piece. At first for the considered model damping was supposed that proportional damping model be with linear combination of mass and stiffness matrix. The relation of this proportional damping is offered in equation (1).

$$[C] = A[K] + B[M] \qquad (1)$$

This optimization is done graphically. In this way that by the change of A and B values in a range the norm of difference among damping values is determined. By drawing the difference according to A and B the minimum point, show the optimized value of these parameters. The coefficients that have produced the least error have been selected as optimized coefficients. The values of damping coefficients resulted from test and corrected model have been shown in the table (4).

Table 4: The values of damping coefficients resulted from test and corrected model

Mode number	ʒ1	ʒ2	ʒ3	ʒ4
Updated	0.0240	0.0141	0.0199	0.0104
Experimental	0.0248	0.0138	0.0208	0.0090
Error (%)	3.2258	-1.2097	3.6290	-5.6452

In continuation, in order to evaluate resulted finite element model, frequency response functions diagrams resulted from test are reproduced and compared by corrected finite element model that have been shown in the figure (1).

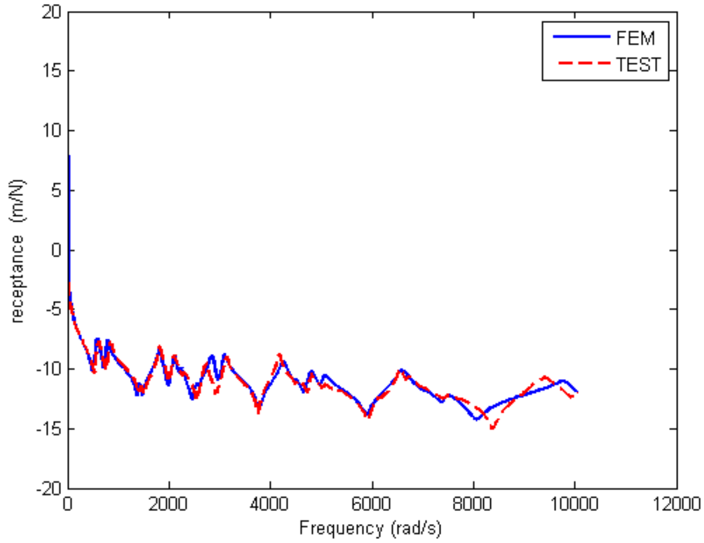

Figure 1: The comparison of total frequency response functions diagrams of board with a TQFP piece

DYNAMIC STRESSES CALCULATION

Dynamic Stresses Calculation in Solder Joints of Finite element Model of Board with a TQFP Piece

In the previous sections, the corrected finite element model with the test results for discussed electronic board in this article obtained. The obtained model has the capability of simulation dynamic behaviour of real board from natural frequencies point of view and from damping coefficients point of view with acceptable exactness. Damping matrix (proportional) in addition to mass and stiffness matrixes obtained for the model. In this section, the aim of using obtained finite element model is to determine the stresses in solder joints. At first by using equation (2) frequency response functions matrix of updated finite element model of board with a TQFP piece is calculated.

$$H(\omega) = [K - \omega^2 M + j\omega c]^{-1} \tag{2}$$

For calculating the stress in the joints, it is necessary to get the displacement of the springs two sides. To get the beginning and end sides displacement, the base of TQFP piece solder joint and PCB board that have been modeled linearly, the equation (3) is used.

$$x_i(\omega) = H_{ij}(\omega) \times f_i(\omega) \tag{3}$$

In this equation I is the response point and j the point of applying fj excitation. Given that the beginning and final points of joint base that has simulated with linear spring in the z direction be p and q the stress equation will be as equation (4).

$$\sigma(\omega) = \frac{1}{As} K_z \left(xp(\omega) - xq(\omega) \right) \tag{4}$$

Which in the equation Kz is the stiffness coefficient model spring in z direction and As is the equivalent section area of solder joint. By referring to the standards of TQFP electronic pieces that is offered in the appendix the area of solder joint obtained 0.00000025 mm^2. The appendix for instance the technical properties of the TQFP piece that it has 144 pine and made by Altera (Altera, 2004) Company has been offered. The Kz coefficient obtained from updating 209 Newton per meter was considered. In the case of that applied acceleration on the board be in the frequency domain a(w) and mass of board m, ma static force is entered on all points of board. Therefore, excitation force in the frequency domain is calculated from equation (5).

$$F(\omega) = \begin{Bmatrix} f_1(\omega) \\ 0 \\ 0 \\ f_2(\omega) \\ \vdots \\ f_n(\omega) \\ 0 \\ 0 \end{Bmatrix} = \frac{ma(\omega)}{n} \begin{Bmatrix} 1 \\ 0 \\ 0 \\ 1 \\ \vdots \\ 1 \\ 0 \\ 0 \end{Bmatrix} \tag{5}$$

By replacing equation (5) in equation (3) and using super position principal can obtain the displacement of p and q points. The displacement of p point obtains from equation (6).

$$x_p = \frac{ma(\omega)}{n} \begin{bmatrix} H_{p1}(\omega) & H_{p2}(\omega) & \ldots H_{pn}(\omega) \end{bmatrix} \begin{Bmatrix} 1 \\ 0 \\ 0 \\ 1 \\ \vdots \\ 1 \\ 0 \\ 0 \end{Bmatrix} \tag{6}$$

As was mentioned previously, m in the equation (5) and (6) shows the total model mass of board with TQFP piece that equals 95 gr. From this equation the displacement of beginning and final points of spring which its corresponding nodes are respectively 691 and 2140, are calculated, n in this equation is the number of finite element model nodes of board with a TQFP piece namely 729 nodes. For response's calculation in the q point said issues is true, too. Equation 6 calculates the response in the q point. To calculate acceleration a(w) it is used of standard excitation of Boeing Company (Tom, Vic, 2010). Input excitation in this test is as power spectral density in different levels of acceleration. In figure (2), the diagrams of input power spectral density (PSD) in different levels to the model board with an updated TQFP piece have been offered. To get excitation input acceleration equation (7) is used. In this equation, PSD and related frequency obtain from figure (2) and finally can get excitation input acceleration in different levels.

$$G_{in} = \sqrt{PSD \times Hz} \tag{7}$$

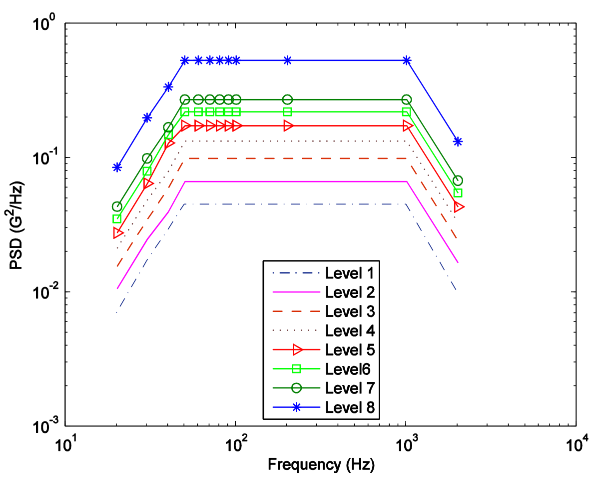

Figure 2: PSD diagram in 8 input excitation level

By determining, the displacement between two sides of spring from equation (4) can obtain the produced dynamic stress in springs in the frequency domain. By using of Fourier conversion reverse dynamic stresses in the time domain are calculated, equation (8) offers dynamic equation in the time domain.

$$\sigma(t) = \int_{-\infty}^{\infty} \sigma_{(\omega)} e^{j\omega t} d\omega \qquad (8)$$

Inspecting TQFP Solder Joint failure in Different Excitation Levels

In this section at first in each level of input applied acceleration, the diagram of excitation force in the time domain and in continuation the stress diagram in the frequency domain and time is offered. At final, in the table of comparison between maximum stresses in each level of acceleration with solder joint yield stress have been offered. The temperature in this joint yield stress analysis was in 25 centigrade degree environment temperature equal to 29.1 Mpa (Kinyanjui, Jabil, Quyen). Solder. In figure (3) the diagram of stress in time domain has been shown for eighth level. Table (5) shows maximum dynamic stress of solder joints in different excitation levels.

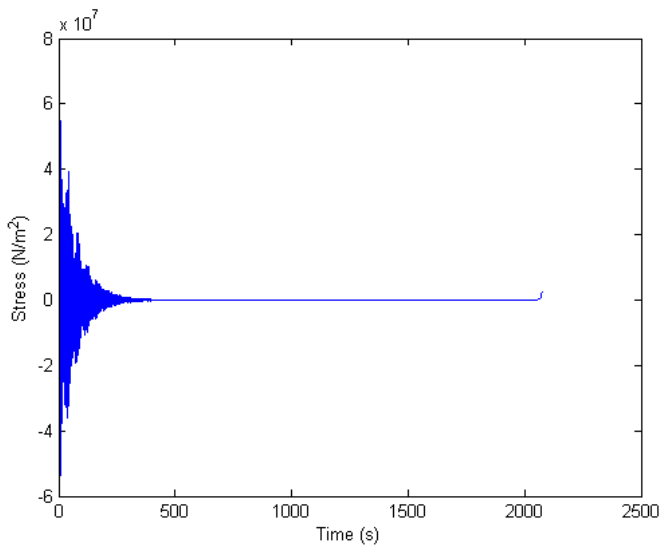

Figure 3: Stress diagram in time domain for acceleration eighth level

Table 5: Comparing maximum dynamic stress obtained in joint with Sn60-Pb40 joint yield stress in 25 C^o

Excitation level number	Maximum dynamic stress obtained (Mpa)	Joint yield stress (Mpa)	Joint situation
1	18.44	29.1	OK
2	22.49	29.1	OK
3	27.48	29.1	OK
4	31.81	29.1	failed
5	36.34	29.1	failed
6	40.89	29.1	failed
7	45.43	29.1	failed
8	63.61	29.1	failed

CONCLUSION

With regard to high sensitivity of electronic pieces in aerospace structures functionally, it is necessary to pay special attention to inspect entered loads and calculating entered stresses to these pieces especially to joints. Because, in comparison with other components of aerospace structures being assure of proper performance of these pieces during aerospace mission of the structure and preventing these piece failure is very important issue. This article allows the designer of aerospace structures by modeling vibration different loading conditions can investigate the electronic pieces performance in each stage of structure aerospace missions and at final do proper designation. Having a proper finite element model and extracting dynamic parameters of these boards and being aware of joints dynamic stresses make the designation of these pieces in real conditions possible. As was in different dynamic conditions is very critical. This was mentioned the main failures in aerospace structures are resulted from dynamic loading. Regarding the importance of aerospace structures mission being aware of the manner of these pieces response this research offers a model with least error that can be a base for structure designation. By having this updated model and changing its parameters can offer optimized designation for more complex models because used method for each complex piece is usable, too.

REFERENCES

Altera., Max II Device Handbook. (2004). Volume 1 "Section II. PCB Layut Guidelines". Chapter 7,8,9.

Ewins ,D.J. (2000). Modal Testing: Theory and Practice. *Research Studies*. John Wiley Press Ltd.

John H Lau and Rice W. Donald. (1985). Solder joint fatigue in surface mount technology. State of the art. *Solid State Technology* 91–104.

Ju, S. H., Sandor, B. I., & Plesha, M. E. (1996). Life prediction of solder joints by damage and fracture mechanics. *Journal of Electronic Packaging* **118**(4) 193-200.

Kinyanjui, Robert.,Jabil,Chu., Quyen. Solder Joint Reliability Of Pb-Free Sn-Ag-Cu Ball Grid Array(BGA) Component in Sn-Pb Assembly Process.

Lau, J., Powers, L., Baker, J., Rice, D., & Shaw, B. (1989, September). Solder joint reliability of fine pitch surface mount technology assemblies. In: *Electronic Manufacturing Technology Symposium 1989, Proceedings. Seventh IEEE/CHMT International* (48-60). IEEE.

Luan, J. E., Tee, T. Y., Pek, E., Lim, C. T., & Zhong, Z. (2007). Dynamic responses and solder joint reliability under board level drop test. *Microelectronics Reliability* **47**(2) 450-460.

Pitarresi, J. M., Caletka, D. V., Caldwell, R., & Smith, D. E. (1991). The "Smeared" property technique for the FE vibration analysis of printed circuit cards. *Journal of Electronic Packaging* **113**(3) 250-257.

Steinberg, D. S. (1988). Vibration analysis for electronic equipment. New York, Wiley-Interscience, 1988, 460.

Tin-Lup Wong, Karl K. Stevens, Junshi Wang, and Wayne Chen. (1990). Strength analysis of surface mounted assembles under bending and twisting loads. *Journal of Electronic Packaging*, No.112 168–174.

Woodrow,Tom.,Starkovich,Vic. (2010). NASA-Dod Lead-Free Electronics Project:Vibration Test". Boeing Electronics Materials and Processes Report-603,Rev.A (EM/P-603,Rev.A),C40C-D019-01

Zhang, B., Ding, H., & Sheng, X. (2008). Modal analysis of board-level electronic package. *Microelectronic Engineering* **85**(3) 610-620.