A Time Domain-based Approach for BGA Solder Joint Fatigue Analysis using Global Local Modeling Technique.

The fatigue life prediction of the electronic packages under dynamic loading condition is an increasingly important area of research, with direct application in packaging industries. Current life prediction methodologies are, in general, developed through a Finite element (FE) model that is correlated using an experimental data generated through sweep sine testing. The frequency response curve generated by using a sweep sine testing may suffer from leakage and windowing of the signal may not work correctly which results in the shift in the amplitude and the resonance frequencies of the package. As a results of which, there might be a significant deviation between the actual and the predicted natural frequencies and the amplitude of vibration in the given excitation range which may results in the longer time to fail the package during the lab based /virtual durability testing. Thus, it is necessary to develop the different validation technique in time domain to address this issue.

In this paper, the step sine testing procedure is utilized to validate the FE model of a test vehicle consisting of BGA chip package and the resonance based fatigue testing is performed in the FE based simulation. The global-local modeling approach is utilized to model the test vehicle and the volume average von misses stress is used to predict the life of the solder joint. Following the numerical simulation, fatigue testing has been carried out in the test vehicle at the first resonance frequency obtained from the step sine test. Experimental results show that there are full openings of the corner balls in a very short interval of time. The results of the life prediction from the FE model and from experiments are comparable to each other thus validating the proposed methodology.

1. Introduction

System level electronic assemblies have components assembled using the solder joint. These electronic assemblies during operation are subjected to environmental stress conditions that are in the form of temperature cycling, dynamic loading and humidity. The environmental stress induces the fatigue load in the joints of the components. The fatigue loading degrades the mechanical strength of the solder joint which eventually leads to a failure of the joint. As a result, improving the long-term reliability of electronic assemblies requires a better understanding of fatigue using environmental stress conditions. Cyclic thermal loading, which has been identified as an important source of fatigue failure in electronic assemblies, has been well studied and documented in the existing literature [1-2]. However, the demand for the use of the electronic assemblies in the transport industries such as: Robo taxis, fleeting truck, locomotives and the aircraft, where dynamic environments are extremely harsh, the reliability of the solder joint used in electronic assemblies must be evaluated before the product is placed on the market. The current trend for analyzing fatigue in an electronic package due to dynamic loading follows two tracks: (i) a stress-based approach and (ii) a strain-based approach. Both approaches are based on a combination of experimental and theoretical modelling. The fatigue life model of the electronic package essentially consists of (i) developing the finite element (FE) model, (ii) correlation of FE model using the experimental results (iii) Generation of stress/ strain in the model (iv) utilize the stress / strain to predict the components life during dynamic loading conditions and (v) validate the theoretical modeling results through direct experimentations.

Whether the procedure adopted is a stress life based approach or a strain life based approach, the global local modeling has been extensively used in the existing literature [3, 5, 9, 10, 16, 17 and 20]. The global coarse model is used for the development of the displacement boundary conditions for the local model and the stress distribution in the solder joint is calculated using the local model. To compensate the effect of mesh density, the volume average von-Mises stress in the thin layers of elements of the critical solder joint is calculated [3, 4, 5, 6, 7, 8, 11, 17 and 20]. Although the majority of the mechanical properties of electronic components are well known, updating the model is necessary for a complex model. For a complex model consisting of integrated packages, the FE model is updated by using the response surface method algorithm [5, 12, and 13]. Despite the fatigue life prediction methodology described in the existing literature is developed for harmonic excitation as well as random vibration excitation operating both in the time / frequency domain, the model validation is performed using the experimental sweep sine data [5, 6,7, 8, 16,17,18, and 20] and has been widely popular in the existing literature as a mean of (i) response validation between the FE model and the experiments, (ii) prediction of the natural frequencies of the package under testing, (iii) correlation of natural frequencies between the package and the FE model and (iv) comparison of harmonic fatigue test results from FE model with the experimental results. However, very careful judgement is required when deciding on the direction of the sinusoidal sweep, the rate of sweep, the input acceleration level, the signal windowing and the type of windowing used. The situation will be worse when the package under test is weakly nonlinear (geometric) which leads to a jump during the forward or backward sweep. In other words, different sweep directions lead to different measured values for natural frequencies [7]. As the frequency of the signal continuously changes over time during the sweep test, the amplitude response of the system under test will produce the transient response. Transforming this transient response from the experiments using FFT to generate the frequency response function (FRF) and use the FRF to correlate the FE model of the system will not be accurate, as all commercial FE modeling software generate the steady state amplitude response of the system over the excitation frequency range. An alternative may be the comparison of transient response data from experiments and the FE model by providing the shaker input data as an input to the FE model. In doing so, the computational cost associated with model development is extremely high and generally impracticable. So, a suitable validation algorithm is necessary to accurately predict the fatigue life the solder joint. The approach presented in this paper is based on time domain-based experiments to carry out model validation. The approach use the sinusoidal step signal as an input and compares the steady-state response data of experiments at each excitation frequency with the harmonic response data of the FE model. The authors believe that the approach presented in this paper for the fatigue analysis and validation of the solder joint to be:

(i) Applicable to predict fatigue life for harmonic loads as well as random vibration loads.
(ii) Leakage free, as the response signal from the experiment does not require post-processing.
(iii) Can accommodate a weakly non-linear system.
(iii) Readily understandable to practicing engineers and scientists.

The rest of the paper is organized as follows. Section 2 of this paper provides a comprehensive approach to predict the fatigue life of the solder joint and detailed on the test vehicle. Section 3 presents the results from the experiments and the FE simulation. Section 4 presents the failure analysis and finally Section 5 concludes the paper with the findings of the research work.

2. Methodology to Evaluate the Fatigue life of the Package

In this section, the methodology to evaluate the fatigue life of the BGA package is described in detailed. The methodology consist of FE modeling of the test vehicle, step sine testing in the FE model, validation of the FE model using the step sine test experimental results, development of the local model, estimation of the fatigue life of the solder joint based on the local model and comparison of the fatigue life through direct experimentations. The critical failure location in the package is a solder joint which has the life of more than 10,000 cycles, a stress life approach is taken for the analysis. The methodology starts with some assumed material properties of the test vehicle and the mode shapes and resonance frequencies are evaluated using the FE global model. Based on the resonance frequency of the FE global model, the step sine test experiments is carried out in a test vehicle. The material properties such as: young's modulus of elasticity, density of the test vehicle is updated until the resonance frequency of the test vehicle from the experiment matches with the FE model results. The damping in the FE model is updated by using the step sine test experimental results. Once the FE global model is updated, the local model is utilized for stress calculation by modeling one of the corner joint of the solder ball.

2.1 Test Vehicle:

The test vehicle used in this study consist of BGA chip soldered to the PCB using the lead free (SAC 305) Tin/Silver/Copper alloys that contain 96.5% Tin (Sn), 3% Silver (Ag), and 0.5% Copper (Cu). The CAD model of the test vehicle is shown in Figure 1. Altogether, 769 solder balls are used between the substrate and the PCB. The dimensions of the PCB used are 140mm in length, 140mm in width and 1.6mm thick and the substrate dimensions are 24.5mm in length, 24.5 mm in width and 0.64mm thick. Figure 2 shows the manufactured daisy chain PCBA for vibration testing.

The PCB used is made of FR4 material. There are 8 holes around the corner edges, these holes are used to secure the test vehicle assembly on the fixture plate using standoffs.

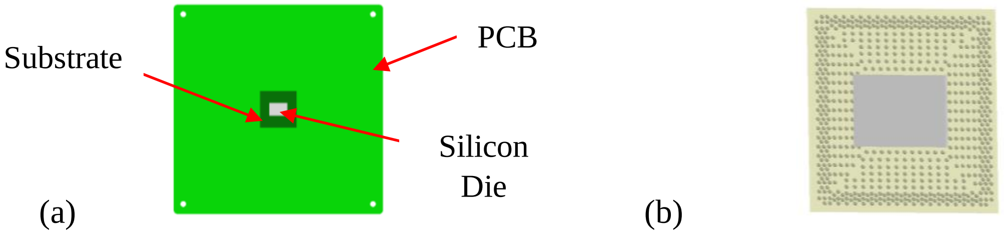

Figure 1: Test Vehicle CAD Model: (a) Board Detail, (b) Solder Ball Detail

Figure 2: Test Vehicle Assembly for Experimental Evaluation

2.2 Finite Element (FE) model

The finite element FE model of the test vehicle is modeled using ANSYS 19.0 Structural analysis tool. For a global model, modeling all the solder ball will increase the complexity of the model. Therefore, only the outer rows and columns of the solder balls, along with the substrate and die are modeled because during dynamic loading, stress induced in the inner balls is minimum. The main step in the FEA modeling is generating a mesh. FEA uses a complicated system of points called nodes that make a grid called a mesh. This mesh is programmed to contain the material and structural properties that define how the structure will respond to particular loading conditions. The use of tetrahedral mesh generates extremely high number of elements resulting the degree of freedom of a system to be very high. So, to minimize the number of elements and decrease the computational cost, all the components in the test vehicle are meshed using quadratic hexahedral elements. Figure 3. Shows the global model of the test vehicle. The model is developed by gluing parts to each other. Figure 4 shows the local cut model of the test vehicle. The local cut model is developed by taking one of the corner solder ball. It should be noted that while creating a cut model, the cut boundary for the local model need to be verified to be sure that it is far enough from the stress concentration region.

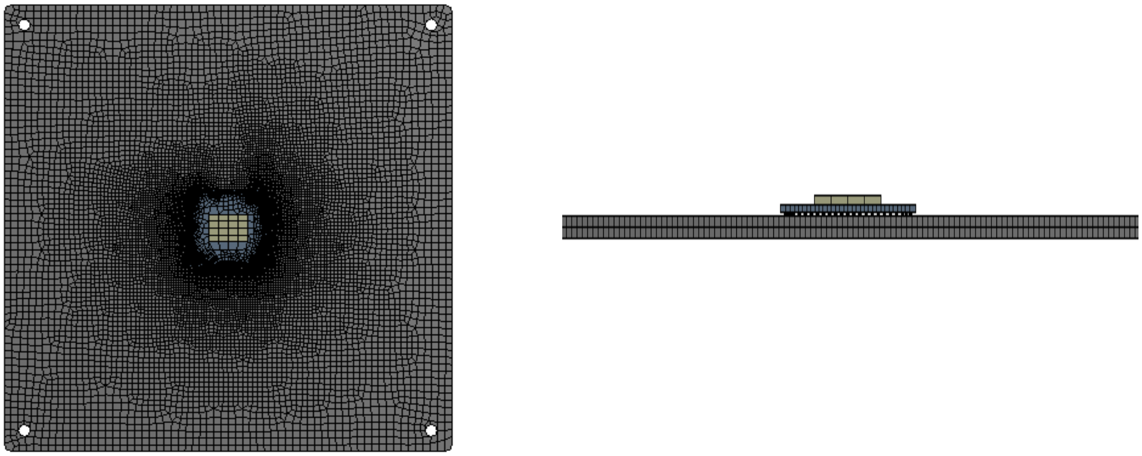

Figure 3: Mesh Model of the test vehicle (Global Model)

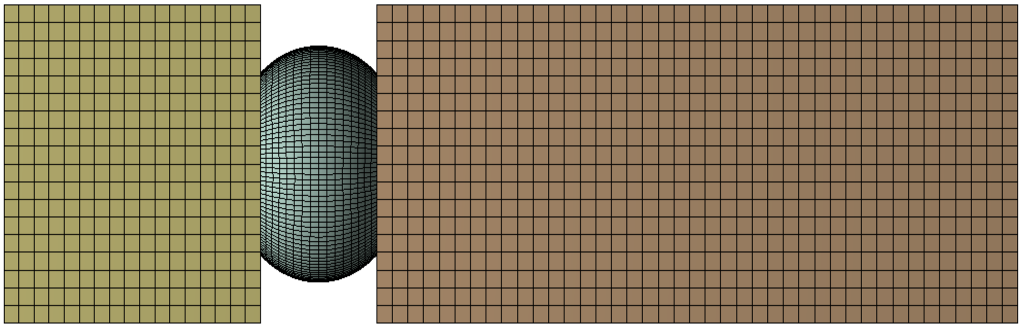

Figure 4: Local Cut Model for Maximum Stress Calculation

2.3 Experimental Set up:

Experiments are carried out in two steps: (i) validation of the amplitude response of the FE model and (ii) Fatigue testing at the first natural frequency of the test vehicle. Preliminary modal analysis using FEA shows that the mass participation factor in a first mode is extremely high and the mode of vibration is bending, fatigue testing is carried out at the first resonance mode of the test vehicle. The experimental set up consist of test vehicle mounted to the electrodynamic shaker head by using the suitable fixture. The fixture is designed in such a way that the natural frequencies of the fixture are far above the excitation frequency range. 4 corner screws with appropriate tightening torque are used to mount the test vehicle into a fixture. The controlled accelerometer is mounted at the center of the fixture to generate the controlled acceleration signal which will be used to excite the test vehicle via 4 corner screws. Figure 5 shows the detailed of the test set up. Accelerometers are used to measure the response of the test vehicle and they are directly mounted at the PCB. The reference location of the accelerometers in the PCB is measured so as to correlate the response at the same location from the FE model.

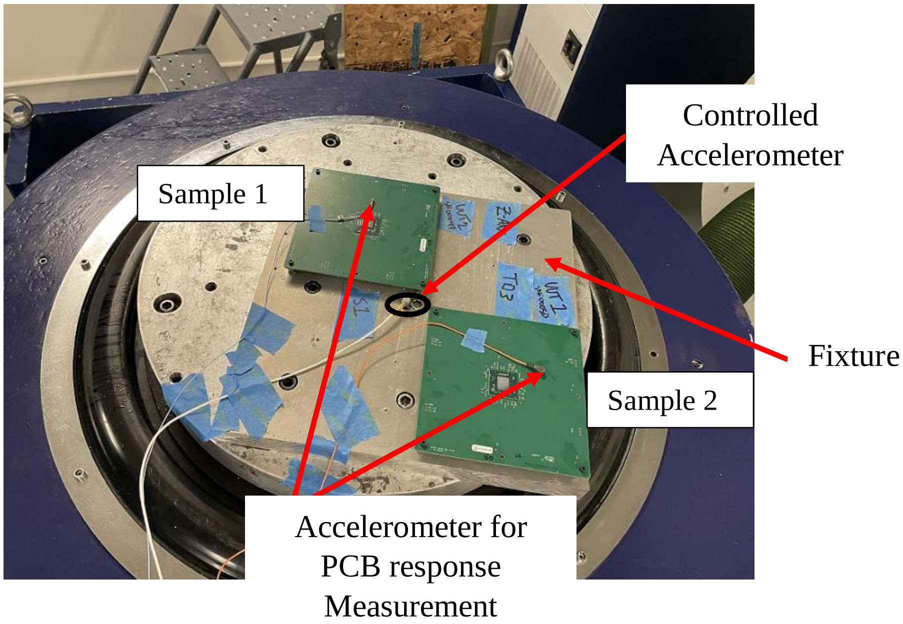

Figure 5: Experimental Set-up

2.4 FE Model Validation:

Modal analysis is carried out for the FE model shown in Figure 3 by assuming the material properties of the PCB and substrate to be FR4 and the properties of SAC 305 and silicon die are used from the existing literature. Although the FR4 material has anisotropic behavior, for simplification FR4 is assumed to be isotropic. Since the FR4 has a wide range of variation of the material properties (density, young's modulus of elasticity), an initial value is assumed and the modal analysis is carried out to calculate the natural frequency and mode shapes of the test vehicle. Based on the natural frequency obtained from finite element model, the step sine testing experiment is carried out to locate the actual natural frequency of the test vehicle. The natural frequency of the test vehicle in the FE model is updated by changing the parameters of the FR4 until the natural frequency is closed to the actual measured frequency of the test vehicle that is measured through step sine testing. Table 1. Shows the final values of the material parameter used for FE model simulation. The modal damping ratio of the test vehicle is estimated by using the half power bandwidth equation shown by Equation (1) and is updated in the FE model. Step sine testing is carried out in the FE model using the ANSYS harmonic analysis tool and the FE model is compared with the experimental step sine test results. The detailed of the step sine testing procedure is explained in Section 2.5.

The half power bandwidth equation for the estimation of damping can be written as,

$$\xi = \frac{\Delta f}{2 f_n} \qquad (1)$$

Where ξ represents the modal damping ratio, Δf is the difference in frequencies corresponding to the half power bandwitdth (-3dB) that is 0.707 of the peak amplitude and f_n corresponds to the frequency at the maximum amplitude of vibration.

2.5 Step Sine Tests

In order to validate the FE model with the experimental results, experiments are carried out on a test vehicle by using the step sine signal. Essentially the step sine signal experimentations work completely in the time domain and no further processing via FFT (Fast Fourier transform) and windowing of the signal is needed. In the step sine signal, the excitation amplitude of the input signal is kept constant and the system is excited in a wide range of frequency bands. At each excitation frequency, the steady state amplitude response at the desired location of the test vehicle is obtained. Finally, a frequency response function is generated from the response amplitude at each excitation frequency. As the step sine testing process is time consuming because steady state response data need to be collected at each excitation frequency, the frequency band of testing can be selected based on: (i) uncorrelated harmonic response of the FE model, (ii) around the natural frequency based on the initial modal testing using the modal hammer/shaker and (iii) around the natural frequency based on the frequency scanning/sweep sine testing using the electrodynamic shaker directly. In this paper, frequency band corresponding to uncorrelated harmonic response of the initial FE model is taken as the test frequencies. Figure 6 shows the step sine testing performed in one of the sample test vehicle. The data points shown in Figure 6, correspond to the steady state relative acceleration of the test vehicle at the measured degree of freedom (DOF) with respect to the base at each excitation frequency. The measured DOF is shown in Figure 5 which is at a distance $(70mm \times 40mm)$ from the test vehicle. The linear equation of motion for a test vehicle subjected to a base excitation can be written as,

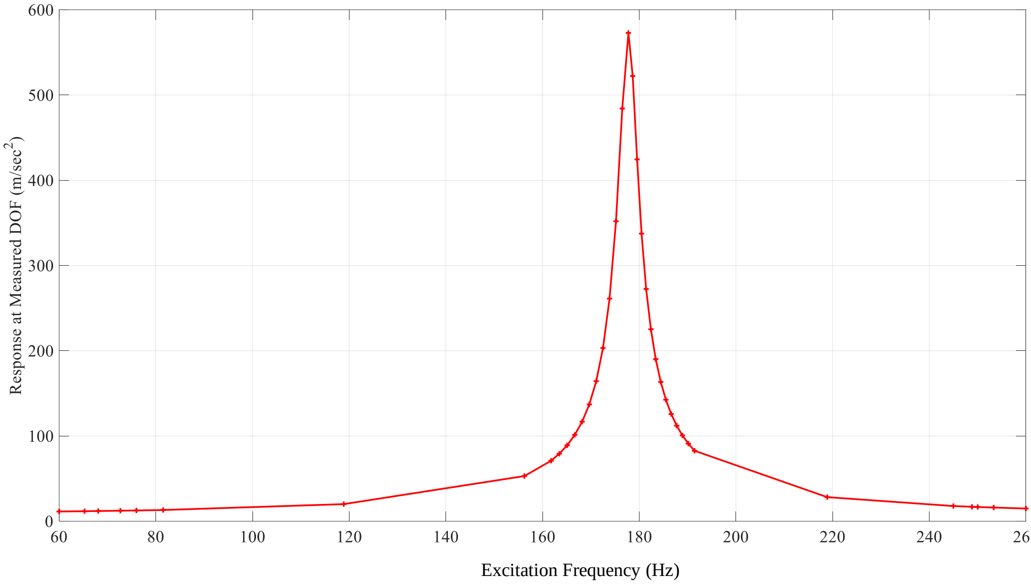

Figure 6: Step Sine Response of the Test Vehicle as a Function of Excitation Frequency

$$[M]\{\ddot{U}\} + [K]\{U\} + [C]\{\dot{U}\} = -[M]\{l\}\{\ddot{Y}\} \tag{2}$$

Where $[M]$, $[K]$ and $[C]$ are the mass, stiffness and the damping matrix of the test vehicle, $\{U\}$ is the relative displacement of the test vehicle at the measured degrees of freedom with respect to the base/fixture motion, $\{l\} = cos\{\theta_i\}$is the transformation vector with $\{\theta_i\}$ representing the angle between the i^{th} degree of freedom and the direction of base motion and $\{\ddot{Y}\}$ represent the controlled acceleration of the fixture plate at the four corner screws. For a harmonic excitation, the solution to Equation (2) is also harmonic. So, if the excitation is made at certain frequency, the steady state response to the system will be at same frequency. Equation (2) can be solved in the commercial FE modeling software by using the harmonic analysis tool/transient analysis tool (The computation cost for harmonic tool will be extremely low as compared to transient analysis). The output of the FE model can be directly compared to experimental step sine results. Figure 7 shows the comparison of the acceleration response between the FE model and the experimental results. The response from the FEA was updated by using the damping ratio estimated from the experimental data using Equation (1) and the material properties for the PCB are varied until the natural frequency matches to the experimentally observed natural frequency. The updated final values of the materials used in FE model are listed in Table 1. A total of, 3 identical samples of test vehicle are tested using the step sine test. There is a small difference in the natural frequencies of each test vehicle sample, the possible reason could be the manufacturing tolerance. Figures (8-9) show the step sine results from two additional test vehicle samples and Figure 10 shows the first mode shape of the test vehicle generated from the FE model.

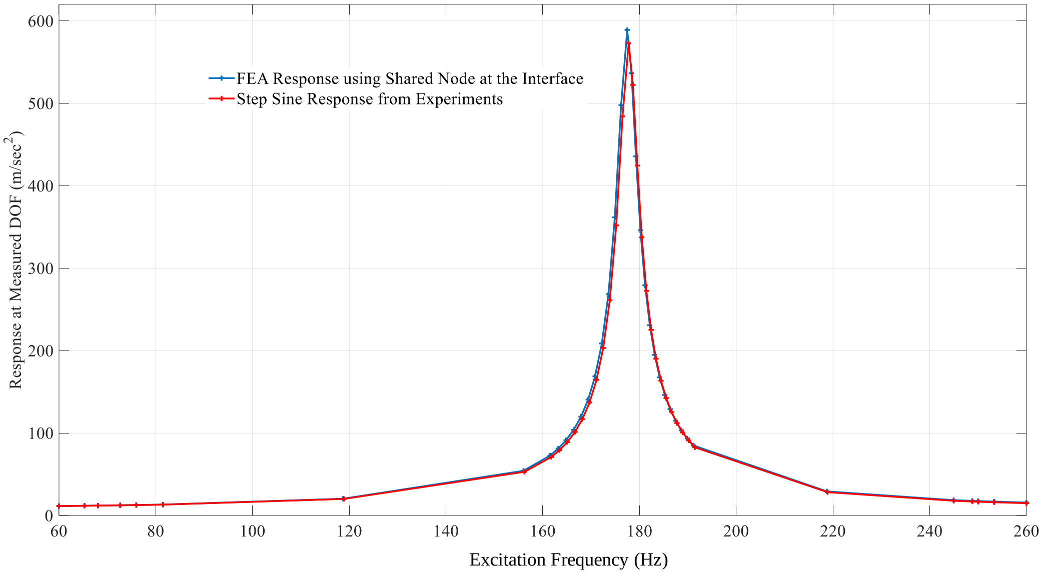

Figure 7: Comparison of Measure and FEA Response of the Test Vehicle

Table 1: Material Properties use in FEA Simulation

Materials	Young’s Modulus (Pa)	Density	Poisson’s Ratio
PCB	1.65E+10	1910	0.39
Substrate	1.65E+10	1910	0.39
SAC 305	5.1E+10	7400	0.3
Silicon	6E+10	2330	0.25

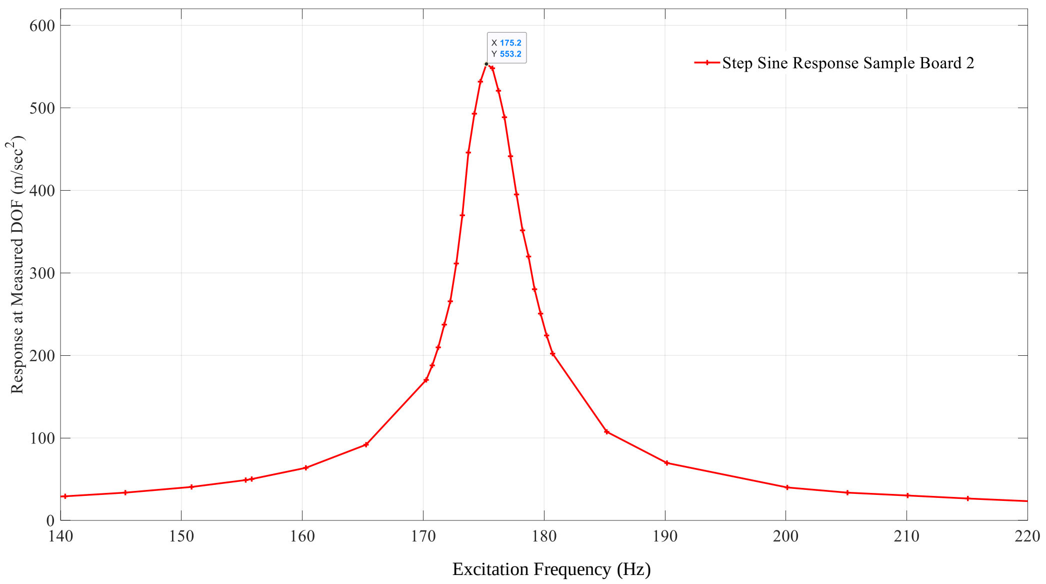

Figure 8: Step Sine Response of the Second Test Vehicle Sample

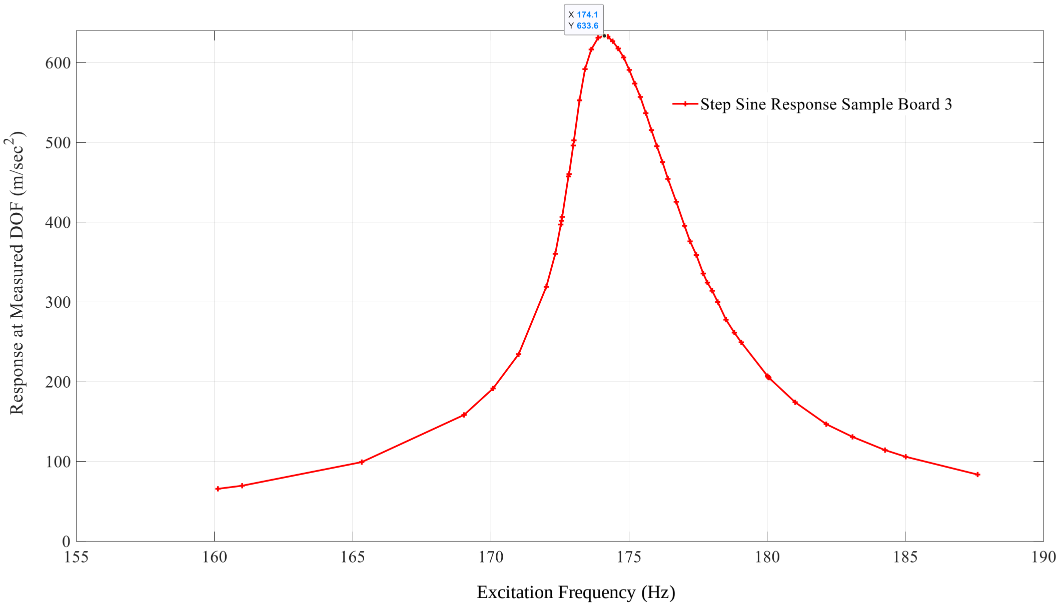

Figure 9: Step Sine Response of the Third Test Vehicle Sample

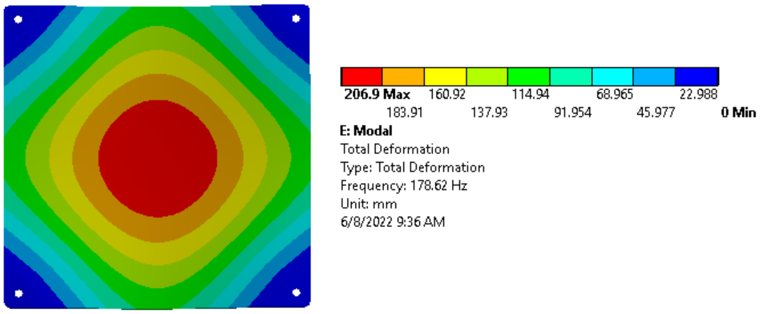

Figure 10: First Bending Mode of the Test Vehicle Generated through FEA

2.6. Fatigue Life Estimation using FE based Model

Once the model is validated using the step sine testing, the next step is to estimate the fatigue life of the Test Vehicle. The method presented here closely follows references [4, 5, 6, 7, 17, 18, and 21] with modifications. In references [4, 6, 7, 8] the harmonic based FE model is developed around a frequency band that cover the natural frequency of the test vehicle and the results from FE model are compared with the fatigue testing based on sweeping the frequency around the resonance frequency of the system. Since the frequency during the testing is continuously changing over the time, the response of the test vehicle at particular excitation frequency may not reached to steady state which results in lower amplitude and the test results may deviate from the steady state harmonic simulation. To overcome this, a dwell sine test fatigue at the first mode resonance frequency is conducted in the simulation as well as in experiments. The global model shown in Figure 3 is excited at an acceleration level of 4G with an excitation frequency of 178.62 Hz (First mode shown in Figure 10) and the steady state displacement response is obtained at the cut boundary. The steady state amplitude at cut boundary is used as the displacement boundary condition for the local model and the corresponding local model is solved for the von-Mises stress. Figure 11 shows the input acceleration to the global model and Figure 12 shows the displacement boundary generated for the local model. As indicated by Figure 12, the relative motion between the package and the PCB is in the order of microns which is the main factor for inducing the maximum von-Mises stress. As the motion of the PCB and substrate are in the same phase, the displacement boundary conditions for the local model is defined by taking the steady state amplitude of displacement given by Figure 12 and the frequency of excitation to be the first mode natural frequency of the test vehicle. This can be written in the form of Equation 3,

$$\{x\} = \{X\} Sin(2 \times pi \times f \times t) \tag{3}$$

Where, $\{X\}$ is the steady state amplitude shown in Figure 12 at cut locations, f is the first mode excitation frequency in Hz ($f = 178.62 Hz$)and t is the time in sec. Careful attention must be taken to select the sampling time to ensure that the response is not distorted. As it is the transient analysis, generating the time series data for a longer time span will increase the computational cost so simulating for (2-3) cycles of steady state response is enough to capture the maximum amplitude of von-Mises stress.

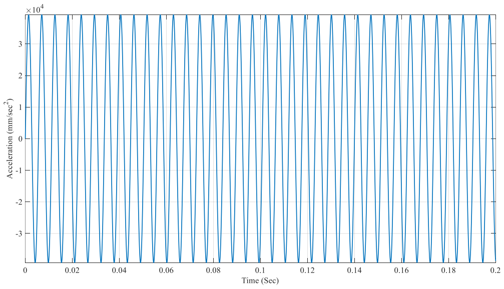

Figure 11: Input Excitation for a Global Model at First Mode Resonance Frequency

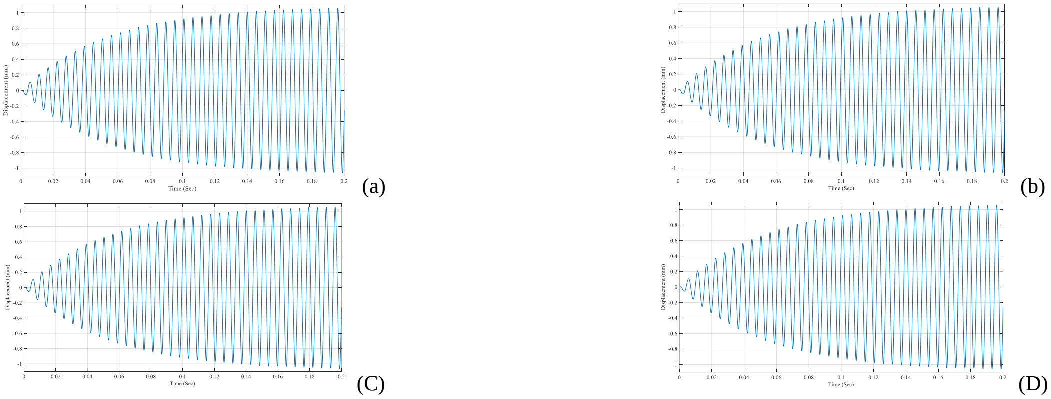

Fig 12: Displacement Response Generated from the Global Model at Cut Boundary (a, b: Substrate side, C, D: PCB side)

3 Results from Simulation and Experiments

In this section, the results from the FE model and experiments are discussed in detail. The time to failure from simulation and experiments are shown and the detailed failure analysis (FA) of the test vehicle is presented.

3.1 FE Model Results

The von-Mises stress in one of the corner solder ball is calculated based on the local analysis of the model. Figure 13 shows the maximum von-Mises stress in the solder joint calculated using the local modeling technique. Maximum amplitude of stress in a solder ball is seen across the package side. A usual procedure to compensate the effect of mesh density is to take the thin layer of elements and calculate the volume average von-Mises stress across that layer [4, 5,6,7,8,9,18 and 21] which require the post processing in FE modeling software. The volume average von-Mises stress is calculated by using Equation 4.

$$\sigma_a = \frac{\sum_{i=1}^{n} \sigma_i \times v_i}{\sum_{i=1}^{n} v_i} \tag{4}$$

Where σ_i is the maximum stress in the i^{th} element, v_i is the volume of the i^{th} element and n is the number of elements in a thin layer across the substrate solder interface. The life of the solder joint is estimated using the high cycle fatigue equation shown by Equation (5).

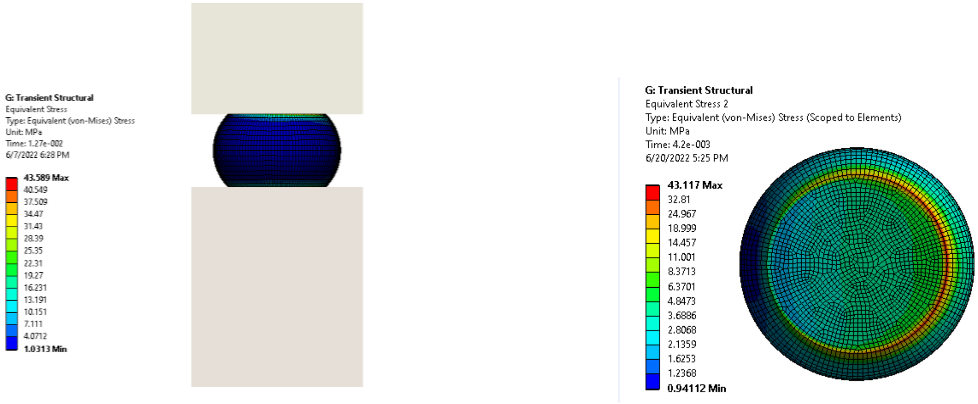

Figure 13: Maximum von-Mises Stress across the Solder Ball

$$\sigma_a = \sigma_f (2N_f)^b \tag{5}$$

Where σ_f is the fatigue strength coefficient, b is the fatigue strength exponent, and $2N_f$ is the number of cycles to failure, the material constant σ_f and b can be estimated using the FE based simulated maximum stress and experiments at several excitation level, several researchers had obtained the material constant using the simulated maximum volume average vonmisses stress [4,6,8]. The life of the solder joint is estimated by utilizing the σ_f and b from the existing literature. The parameters calculated and life of the solder joint estimated from simulation are tabulated in Table 2.

Table 2: Fatigue Test Results Obtained from FE Model

Stress Amplitude (σ_a)	Fatigue Strength Coefficient (σ_f)	Fatigue Strength Exponent (b)	No of Cycles	Time To Failure
9.213MPa	64.8	-0.1443	7.4288E+5	1 Hour, 9 Minutes

3.2 Fatigue Tests (Experimental)

Resonance based fatigue test is carried out in 3 samples of test vehicle at a first resonance frequency with an excitation amplitude of 4G is applied to the test vehicle. Since each test vehicle has slight difference in natural frequency as described earlier, they are tested one at a time. Figure 14 shows the test sample mounted in the Electrodynamic shaker head using the test fixture. As solder joint will fail when the excitation is in the out-of-plane direction, vertical excitation is applied to the test vehicle. Agilent Milli ohmmeter is used to measure the resistance of the daisy chained assembly. The resistance is measured every 15 minutes during the test. According to IEEE standard, when the resistance change is more than 20% than the unit is considered as fail. For all three samples of tested test vehicles, there is no significant change in resistance (less than 5%) until up to 45 minutes of the test. After 45 minutes of testing, the amplitude of vibration measured by the accelerometer continuously started decreasing and all the units failed (resistance increased) after one hour of testing. Figure 15 shows the step sine test results after one hour of fatigue testing which shows the reduction in the natural frequency of the test vehicles. Comparing Figures 6, 8 and 9 to Figure 15, the natural frequency reduction for each sample is in the order of (3-3.5) Hz.

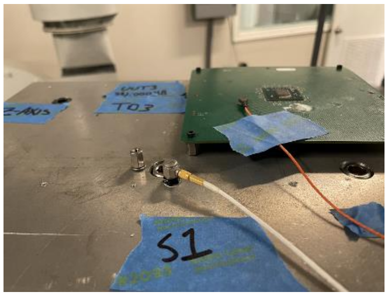

Figure 14: Test Vehicle Mounted to the Shaker Head for Fatigue Testing

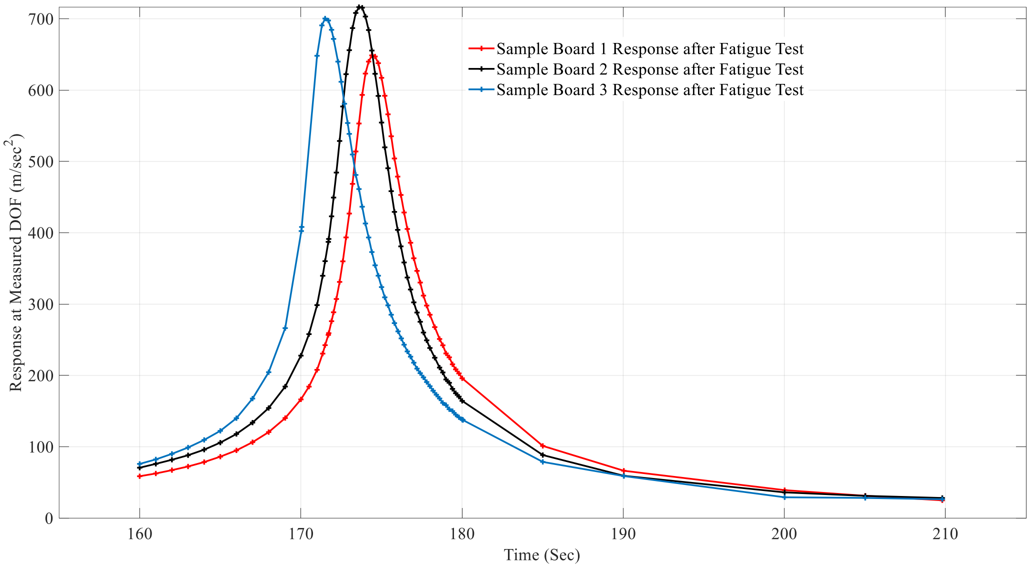

Figure 15: Step Sine Test Results of the Test Vehicle after Post Fatigue Test (Excitation: 1G)

3.3 Failure Analysis

Once the fatigue test is completed, destructive failure analysis in a test vehicle is performed to further confirm the failure. Dye and Pry techniques are applied for failure analysis to confirm the failure mode. Dye and Pry techniques involve submerging in low viscosity red dye and pressurized within an enclosure. A bake is perform to harden the dye prior to the mechanical prying. The dominant failure mode in this study is the solder joint crack at the IMC layer on the package side. (Dongji correct me, if I am wrong). All the corner solder balls are found to be failed. Figures (16-17) show the full length crack of the Solder ball. All the samples tested show the same failure mechanism. As illustrated in the Figures (16-17), a score of C9 and C10 indicates a crack area of 90% and 100% respectively.

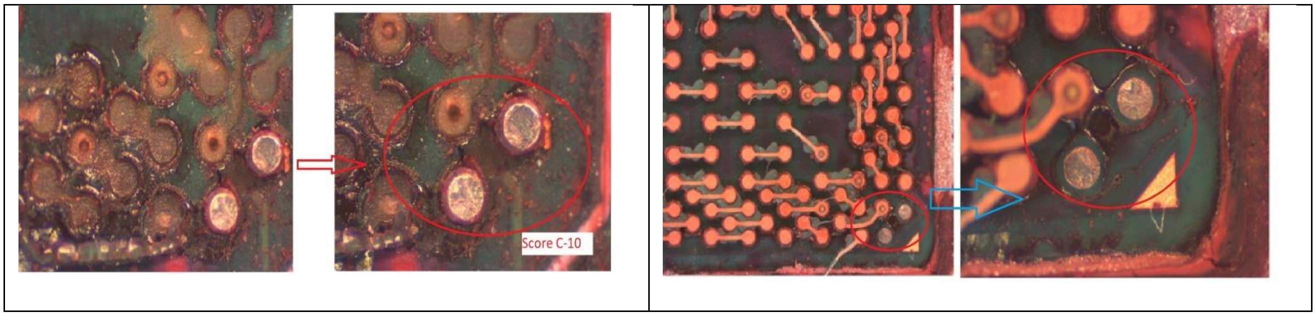

Figure 16: Images of Test Vehicle after Mechanical Prying

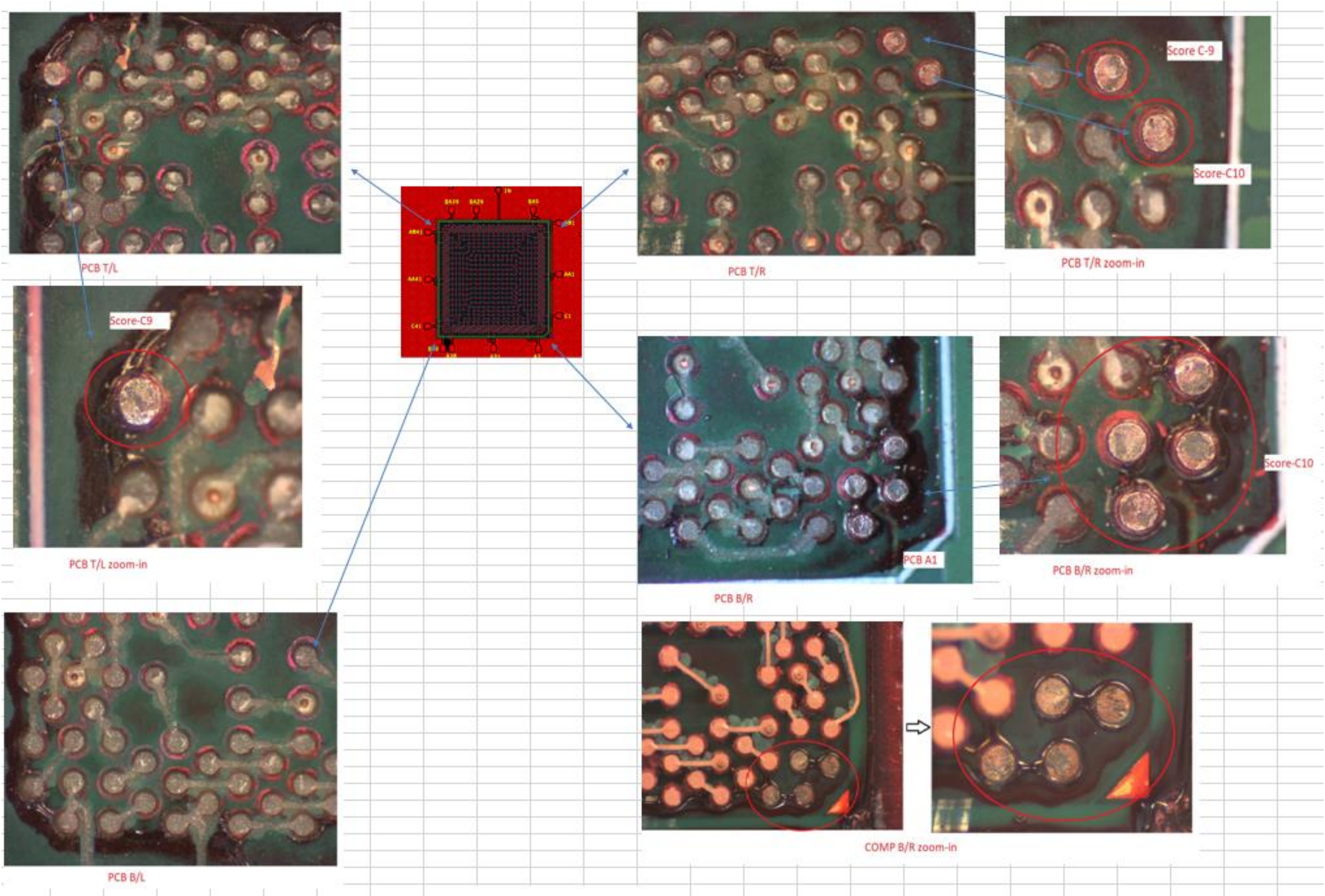

Figure 17: Figure 16: Images of Test Vehicle after Mechanical Prying

4. Conclusion

This paper developed a time domain-based methodology to predict the fatigue life of the solder joint used in the electronic packaging. A step sine test procedure to validate the FE model using the experimental data is presented. Once the model is validated, a procedure for performing the resonance based fatigue testing in a FE based model using time series data is presented. The local global modeling approach is used to calculate the volume average von-Mises stress and the high cycle fatigue equation is utilized to calculate the solder joint life. Following the simulation, fatigue tests are performed in 3 samples of test vehicles at the first resonance frequency at a 4G excitation amplitude. The corner solder balls for all the test vehicles failed at 1 hour which is validated through: (i) the post fatigue step sine test experiments and (ii) conventional dye and pry techniques. The results of the simulation and experimental life tests are close to each other, thus validating the proposed methodology. The specific findings of this research are summarized as follows.

(I) The step sine testing methodology is extremely beneficial for a model validation and can be combined with traditional modal analysis tools.

(II) The step sine method is very effective in predicting the resonance of the test vehicle when there is a jump in the frequency response function that arises from the geometrical nonlinearity of the test vehicle. Moreover, compared to the conventional method, the step sine test does not require the post-processing of the measured signal.

(III) Though the step sine test method is a time consuming test, it can well predict the shift in natural frequency as a result of the solder joint failure.

(IV) The global and local FE model should be developed using the same type of element, modifying the element type can change the displacement response which in turn will affect the maximum stress induced at the solder joint.

(V) The failure of the solder joint depends on the excitation frequency and the first natural frequency is the worst failure frequency. At higher modes, even the cumulative relative movement between the package and the PCB is higher, very high excitation amplitude is needed to fail the joint quickly within a reasonable time frame.

Acknowledgements: The authors would like to acknowledge DynaQual Test Labs, Houston, Texas in providing their labs to perform the step sine experiments and fatigue testing. Special Thanks to Advanced Micro Devices (AMD) for providing the test samples.

References:

1. Ghaffarian, Reza. "Accelerated thermal cycling and failure mechanisms for BGA and CSP assemblies." *J. Electron. Packag.* 122.4 (2000): 335-340.

2. Depiver, Joshua A., Sabuj Mallik, and Emeka H. Amalu. "Thermal fatigue life of ball grid array (BGA) solder joints made from different alloy compositions." *Engineering Failure Analysis* 125 (2021): 105447.

3. Malatkar, Pramod, et al. "Pitfalls an engineer needs to be aware of during vibration testing." *56th Electronic Components and Technology Conference 2006*. IEEE, 2006.

4. Cinar, Yusuf, et al. "Failure mechanism of FBGA solder joints in memory module subjected to harmonic excitation." *Microelectronics Reliability* 52.4 (2012): 735-743.

5. Yu, Da, et al. "High-cycle fatigue life prediction for Pb-free BGA under random vibration loading." *Microelectronics Reliability* 51.3 (2011): 649-656.

6. Xu, Fei, et al. "Fatigue life prediction for PBGA under random vibration using updated finite element models." *Experimental Techniques* 40.5 (2016): 1421-1435.

7. Xia, Jiang, et al. "Fatigue life prediction of Package-on-Package stacking assembly under random vibration loading." *Microelectronics Reliability* 71 (2017): 111-118.

8. Cinar, Yusuf, et al. "Effect of solder pads on the fatigue life of FBGA memory modules under harmonic excitation by using a global–local modeling technique." *Microelectronics Reliability* 53.12 (2013): 2043-2051.

9. Che, F. X., and John HL Pang. "Vibration reliability test and finite element analysis for flip chip solder joints." *Microelectronics reliability* 49.7 (2009): 754-760.

10. Fan, Xuejun, Amarinder Singh Ranouta, and Harpreet Singh Dhiman. "Effects of package level structure and material properties on solder joint reliability under impact loading." *IEEE Transactions on Components, Packaging and Manufacturing Technology* 3.1 (2012): 52-60.

11. Fan, Xuejun, and Amarinder Singh Ranouta. "Finite element modeling of system design and testing conditions for component solder ball reliability under impact." *IEEE Transactions on Components, Packaging and Manufacturing Technology* 2.11 (2012): 1802-1810.

12. Jayesh, S., and Jacob Elias. "Finite element modeling and random vibration analysis of BGA electronic package soldered using lead free solder alloy− Sn-1Cu-1Ni-1Ag." *International Journal for Simulation and Multidisciplinary Design Optimization* 10 (2019):

13. Ren, Wei-Xin, Sheng-En Fang, and Miao-Yi Deng. "Response surface–based finite-element-model updating using structural static responses." *Journal of Engineering Mechanics* 137.4 (2011): 248-257.

14. Ren, Wei-Xin, and Hua-Bing Chen. "Finite element model updating in structural dynamics by using the response surface method." *Engineering structures* 32.8 (2010): 2455-2465.

15. Li, Ron S. "A methodology for fatigue prediction of electronic components under random vibration load." *J. Electron. Packag.* 123.4 (2001): 394-400.

16. Steinberg, Dave S. "Vibration analysis for electronic equipment." (2000).

17. Che, F. X., et al. "Development and assessment of global-local modeling technique used in advanced microelectronic packaging." *2007 International Conference on Thermal, Mechanical and Multi-Physics Simulation Experiments in Microelectronics and Micro-Systems. EuroSime 2007*. IEEE, 2007.

18. Cinar, Yusuf, and Gunhee Jang. "Fatigue life estimation of FBGA memory device under vibration." *Journal of Mechanical Science and Technology* 28.1 (2014): 107-114.

19. Xia, Jiang, et al. "Fatigue life prediction of Package-on-Package stacking assembly under random vibration loading." *Microelectronics Reliability* 71 (2017): 111-118.

20. Xia, Jiang, et al. "Comparison of fatigue life prediction methods for solder joints under random vibration loading." *Microelectronics Reliability* 95 (2019): 58-64.

21. Wu, Mei-Ling. "Vibration-induced fatigue life estimation of ball grid array packaging." *Journal of Micromechanics and Microengineering* 19.6 (2009): 065005.

22. Che, F. X., and John HL Pang. "Vibration reliability test and finite element analysis for flip chip solder joints." *Microelectronics reliability* 49.7 (2009): 754-760.