# Development and Assessment of Global-Local Modeling Technique Used in Advanced Microelectronic Packaging

F. X. Che*[1], H.L.J. Pang [2], W. H. Zhu[1], Wei Sun[1], Anthony Y.S. Sun[1] and C.K. Wang[1] and H.B. Tan[1]
[1] United Test & Assembly Center Ltd. (UTAC)
5 Serangoon North Ave 5, Singapore 554916
*Email: FX_Che@sg.utacgroup.com, Tel: 65511465
[2] School of Mechanical and Aerospace Engineering, Nanyang Technological University
50 Nanyang Avenue, Singapore 639798

**Abstract**

In this study, two types of global-local models are introduced. One is submodeling, in which a coarse global model is used to simulate the whole model and the fine local model is used to simulate the critical region of interest from whole model. The other is global-local-beam (GLB) model, in which the joint is replaced by an equivalent beam with effective stiffness. For submodeling, two different cut boundaries are compared and suitable cut boundary is proposed. In addition, the effective global-local model combining submodeling and GLB modeling technique is also introduced. Case study is presented in this paper. Firstly, PBGA assembly subjected to thermal cycling was investigated using global-local FEA modeling. Secondly, the GLB modeling was used in modal analysis for FCOB assembly. Thirdly, bending simulation was conducted for VQFN assembly using both two-level submodeling and one-level submodeling method.

## 1. Introduction

Manufacturers of electronic products face with demands for design with high reliability and performance at lower costs. Computational modeling can reduce the product development time to market in a competitive electronic product sector. Commercial finite element analysis (FEA) software, such as ANSYS, has been used extensively to simulate reliability test loads on electronic components, board assemblies and product systems. By using FEA simulations, industry can minimize the requirement for extensive and time-consuming physical testing. This can reduce product development costs, increase reliability and reduce the product time to market. Further miniaturizations of IC component size and higher I/O counts are expected trends in electronic packaging applications. Thus, conventional application of finite element modeling technique will become more difficult as the geometry features become smaller and require higher number of elements so that FEA simulation will require higher speed computing, larger memory size and hard disk storage space. These critical requirements can limit the use of full 3D model applications for finite element reliability analysis of the electronic assemblies. In order to reduce the element size in the FEA simulation for solder joint reliability, some reduced models were used by researchers, including slice model [1-4], one-eighth model [1-2, 4-5], and 2D model [1, 3-4]. Some trade-off in accuracy is expected in these simple models. The comparison details for different FEA models were presented in reference [1]. In this study, a global-local modeling technique was developed. The global-local modeling method with coarse global model and fine submodel was reported by researchers [6-8] for BGA assemblies subjected to thermal or mechanical loadings. Some simplified models for stress-strain analysis of BGA assemblies subjected to thermal cycling or vibration loads, where the solder connections were considered as an effective beam element with similar stiffness, were employed by researchers [9-10].

Case study is presented using developed global-local modeling technique considering different package types and different loading types. Firstly, fatigue life prediction for PBGA assembly subjected to thermal cycling was investigated using submodeling and effective global-local model. Secondly, the GLB modeling was used in modal analysis for FCOB assembly. Thirdly, cyclic bending simulation was conducted for VQFN assembly using both two-level submodeling and one-level submodeling. For two-level submodeling, the first nodal results transfer is done from board-level global model to package-level submodel, then the second nodal results transfer is done from package-level model to solder joint level submodel.

## 2. Global-Local Modeling Techniques

The global-local model is a good choice for board-level simulation. In this study, two types of global-local models were introduced. One is submodeling, in which a coarse global model was used to simulate the whole model and the fine local model was used to simulate the critical partial area of the whole model. The other is global-local-beam (GLB) model, in which the interconnections were replaced by an equivalent beam with effective stiffness. The results from global-local model simulation were compared to those from fine 3D model simulation for calibration of global-local modeling method.

### *2.1 Submodeling Method*

Submodeling is also known as the cut-boundary displacement method or the specified boundary displacement method. In this study, submodeling was developed using the flip chip on board (FCOB) assembly as FEA calibration study. The geometry size mismatch among FCOB assemblies gives rise to more difficulties in numerical modeling. So the submodeling technique is a good choice to reduce the model size significantly. Before

*1-4244-1106-8/07/$25.00 ©2007 IEEE*

using submodeling technology, the verification was first conducted to validate the feasibility and accuracy of submodeling as well as the appropriate cut boundary. The silicon chip or die was connected onto PCB by four corner solder joints as shown in Fig. 1 for benchmark study in order to reduce element size. The chip size is 8.5mm×8.5mm×0.65mm. For calibration study, the material of 62Sn36Pb2Ag was selected for solder joint with a standoff height of 0.1mm. The FR-4 PCB has the size of 30mm×10mm×1.13mm. In order to determine the underfill effect on the fatigue life of the electronic product, two cases are studied including FCOB assembly with and without underfill between the die and PCB. For convenience, copper pad and UBM material used in actual FCOB assembly are not considered because of just validation study for submodeling technique. When using the submodel, it is necessary to verify that the cut boundaries are far enough away from the stress concentration region, or interest area.

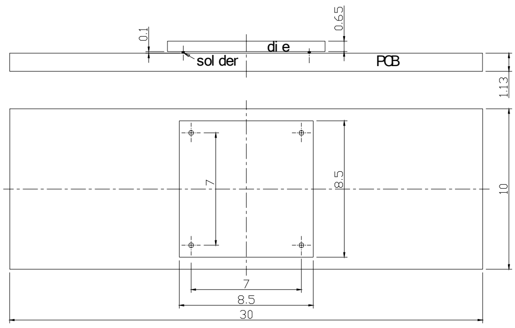
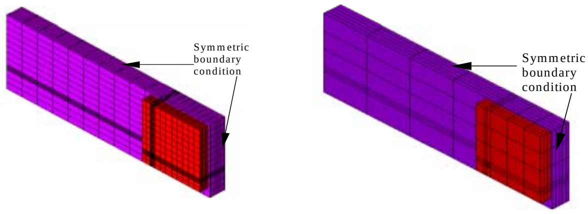


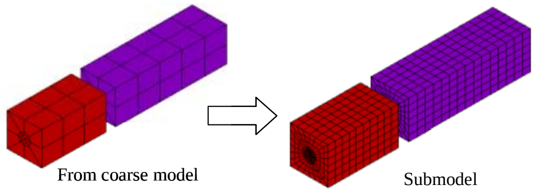

Fig. 1 Geometry size of the FCOB (unit in mm)

Firstly, the main purpose of validation study was to find the reasonable cut boundary and compare the results between the submodel and conventional fine 3D model. The fine 3D model was considered as a reference model, or benchmark model. When the results from submodel agreed with those from reference model, the submodel and corresponding cut boundary were regarded satisfactory. The temperature increment of 20°C was applied to the FCOB assembly as a loading condition. Due to the symmetry, only the quarter part of the assembly was modeled. The 3D models are shown in Fig. 2. Two types of boundary condition as shown in Fig. 3 are conducted to determine the reasonable cut boundary. Fig. 3(a) shows hybrid submodel with a cut boundary far enough away from solder joint interfaces and Fig. 3(b) shows a submodel of solder ball. The grid size of the submodel is same as that of the fine 3D model as shown in Fig. 2(a). The results from submodel were compared with those from the fine 3D model. The Von Mises stress was compared due to its common application in the reliability analysis of the electronic assembly. Figure 4 shows the maximum von Mises stress of solder/die interface comparison between fine 3D model and submodel with two different cut boundaries. It can be seen the first cut boundary gives more accurate result than the second one compared to reference model result.

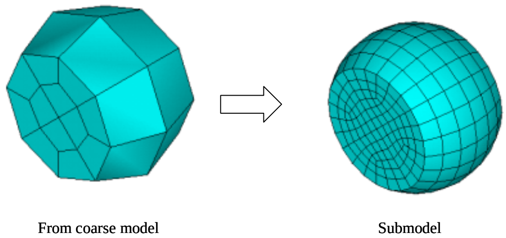



(a) Fine model (b) Coarse model

Fig. 2 Different 3D FE models




(a) Hybrid submodel (cb1)




(b) Solder ball submodel (cb2)

Fig. 3 Two cut boundaries for different submodels

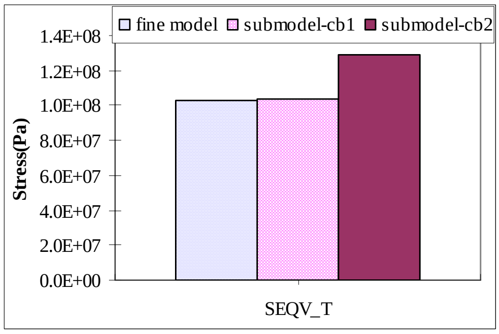



Fig. 4 Von Mises stress for different models

Then, the reasonable cut boundary including die, solder joint and PCB was selected to do the simulation for FCOB specimen when subjected to thermal cycling from –40°C to 125°C with 1hour per cycle and 15mins dwell time. Two cases were considered, that is, FCOB without underfill case and FCOB with underfill case. The elastic-plastic-creep (EPC) model and viscoplastic Anand model are commonly used constitutive models for solder material. These two models can lead to consistent result when thermal cycling loading is simulated [11]. The Anand model was used in this study. The plastic work density is an important parameter in the fatigue life prediction of solder joint when subjected to thermal cycling. Therefore, the plastic work density of center node on die/solder interface was extracted to take comparison and was shown in Fig. 5 for nonunderfill case and Fig. 6 for underfill case.

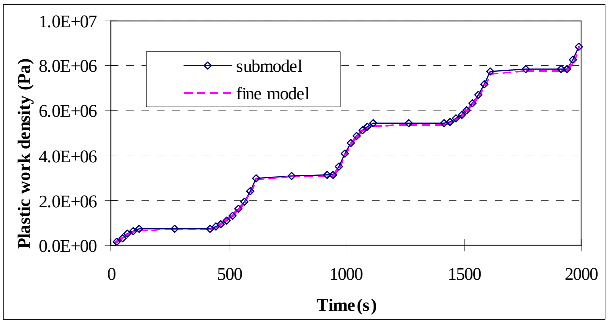



Fig. 5 Plastic work density for nonunderfill case

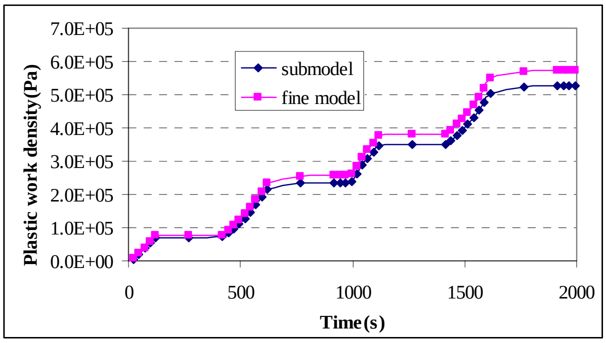



Fig. 6 Plastic work density for underfill case

The error of results obtained from submodeling technology is less than 10% compared with conventional fine 3D model. The advantages of submodeling technique include computational time saving, the hard disc space saving, less memory requirement, and less element size, which can be found from Table 1. When the solid FE model is more complicated, the advantage of submodeling method will become more significant.

Table 1. Advantages of submodel compared to fine model

| Items | | Elements | Nodes | Solving time (Mins) | Result file (MB) |
|---|---|---|---|---|---|
| Sub-modeling | Coarse model (1) | 845 | 1144 | 12 | 113 |
| | Submodel (2) | 5580 | 6176 | 84 | 650 |
| Fine 3D model (3) | | 22905 | 25408 | 671 | 2730 |
| Factor (3)/(1+2) | | 3.6 | 3.5 | 7.0 | 3.6 |

### *2.2 Global-Local Beam (GLB) Model Technique*

Vibration and drop analyses need a full model of the assembly because of non-symmetry in out-of-plane displacement. A full detailed model of BGA or FCOB assembly, which includes numerous solder joints and many modules, is difficult to model. The submodeling technique mentioned earlier can reduce the FE mesh complexity, but is still too complex when many components need to be modeled. Therefore, a global-local-beam (GLB) technique was introduced in this study. The key technique in this method is the use of simple elements, such as beam elements with effective stiffness matrix, to represent solder joints, thus the number of elements needed to model the entire structure can be reduced significantly. This method consists of three steps: stiffness extraction analysis of a single solder joint; deformation analysis of the entire structure and stress strain analysis of solder joint. The specimen selected is same as that shown in Fig. 1. The solder ball is replaced by two-node beam element with effective stiffness and 12 DOFs as shown in Fig. 7. The relation between the generalized nodal force vector $\{F\}$ and the generalized nodal displacement vector $\{u\}$ of beam is expressed in matrix form as follows:

$$\{F\} = \{K\}\{u\} \quad (1)$$

$$\{F\} = [F_{x1} \;\; F_{y1} \;\; F_{z1} \;\; M_{x1} \;\; M_{y1} \;\; M_{z1} \;\; F_{x2} \;\; F_{y2} \;\; F_{z2} \;\; M_{x2} \;\; M_{y2} \;\; M_{z2}]^T \quad (2)$$

$$\{u\} = [u_{x1} \;\; u_{y1} \;\; u_{z1} \;\; \theta_{x1} \;\; \theta_{y1} \;\; \theta_{z1} \;\; u_{x2} \;\; u_{y2} \;\; u_{z2} \;\; \theta_{x2} \;\; \theta_{y2} \;\; \theta_{z2}]^T \quad (3)$$

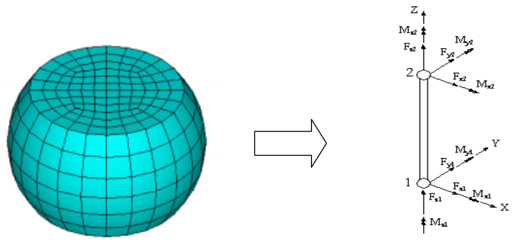


Fig. 7 Solder ball vs. two-node beam element

In the first step, In order to extract the stiffness of the solder joint, unit translational and unit rotational are applied on the surface of the solder joint as a boundary condition. From the solutions the generalized forces imposed at two ends surfaces can be obtained, and these generalized forces form some elements in the stiffness matrix of the solder joint. After 12 different boundary value problems with unit displacement were solved, all the terms in the stiffness matrix can be obtained. When the whole stiffness matrix is obtained, the second step can be simulated. In this step, the solder joints, which connect the chip to the PCB, are modeled as two-node beam elements with the effective stiffness matrix established in the first step. In ANSYS software [12], the element type of MATRIX27 represents an arbitrary two-node element whose geometry is undefined but its elastic kinematic response can be specified by stiffness coefficients. The stiffness matrix constants were input as real constants in the simulation. The chip and PCB can be modeled as solid or shell elements. In this study, these two types of elements are used to model chip and PCB in order to find which one is more accurate. The temperature ranges of 20°C is selected as loading to validate the feasibility of global-local beam modeling method. From deformation analysis, the weakest solder joint should be selected for next analysis. Usually, the relative deformations between two nodes of the beam element are used as an index to determine the weakest solder joint. After determining the weakest solder joint, stress strain analysis of solder joint can be performed. In this step, the nodal displacements at both ends of the beam element are applied to the detailed solder ball model as a prescribed displacement boundary condition. The translational displacements are applied directly at the nodes located on the two end surfaces of solder joint. The rotations are modified to the translational displacements considering the location of each node on the surface firstly, and then the modified translational displacements are imposed on the corresponding nodes. In FEA simulation, the shell element is usually represented by the middle surface of the structure. In the electronic product, the PCB and chip cannot be assumed as the perfect thin shell element because thickness of PCB and IC chip usually do not satisfy the assumption of the thin shell. Therefore, the thick shell element should be considered in the GLB method. According to the theory of thick plate element, the displacements parallel to the middle surface are given by [13]:

$$u(x,y,z) = z\theta_y(x,y) \qquad (4)$$
$$v(x,y,z) = z\theta_x(x,y)$$

where $\theta_x$ , $\theta_y$ are the rotations about the x and y axes, z is the coordinate in thickness direction. It is called the modified shell model when Eq. (4) was used in DOF transfer.

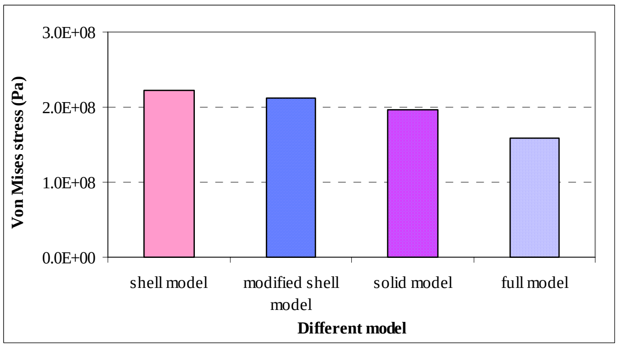



Fig. 8 Von Mise stress comparison for different models

Figure 8 shows the comparison of maximum Von Mises stress in the solder joint from different models. It can be seen that solid model used in GLB model gives rise to more accurate result than shell model. The difference induced by GLB solid model is about 25% compared to the fine 3D model. The difference induced by GLB shell model is larger than GLB solid model, especially for non-modified GLB shell model. The modified shell model is recommended when shell element is used in the entir structure deformation analysis.

### *2.3 Effective Solder Joint Model*

Through analyses for submodeling and GLB model technique mentioned above, it was known that submodeling could lead to more accurate results than GLB model but with more element size and computational resources. It is difficult for GLB model to consider plastic behavior of solder ball because the stiffness of effective beam model for solder joint just considers the elastic deformation of solder ball when extracting stiffness matrix of solder ball. In order to reduce element size and consider the plastic deformation behavior of solder material, an effective solder joint modeling method was developed by combining submodeling and GLB modeling techniques. In effective solder joint model, the ball shape solder joint was replaced by an effective hexahedron shape solder joint with cubic cross section and the same height as solder ball as shown in Fig. 9. According to GLB modeling technique, the cross section area of effective solder joint can be determined by making solder ball and effective solder joint having the equivalent stiffness in axial tension/compression and shear direction because tension/compression and shear forces are dominant for solder joint when subjected to thermomechanical or mechanical loadings.

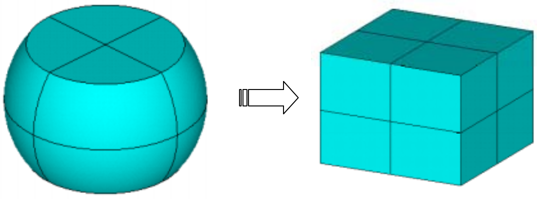


Fig. 9 Real solder ball and its effective solder joint

In the global model, the effective solder joint can be used with fewer elements and plastic deformation can also be considered by an effective solder joint. In order to verify the effective solder joint model, result comparison between effective solder joint model and fine 3D model were conducted considering two cases: FCOB with underfill and without underfill subjected to thermal cycling from –40°C to 125°C. Accumulated plastic work density per cycle based on solder/die interface volume averaged method is shown in Fig. 10. It can be seen that effective solder joint model can lead to consistent result with submodel and fine 3D model for both underfill and nonunderfill cases. The practical use of effective solder joint model is that all the ball shape solder joints are modeled as effective solder joints with cubic cross section except the critical solder joint for which the detailed ball shape meshing was used, and then submodel is created and simulated based on the critical solder joint.

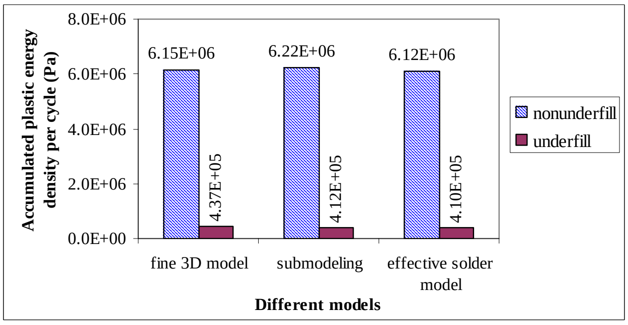



Fig. 10 Accumulated plastic work density per cycle for different models

## 3. Cases Study

### *3.1 PBGA Assembly Subjected to Thermal Cycling*

PBGA specimen was selected for submodeling application case study. The details for thermal reliability test and failure analysis, material properties and loading condition used in FEA modeling and simulation can be referred to the publications [1, 14]. Figure 11 shows FEA models for PBGA assembly including quarter global model and submodel, octant global model and submodel, slice and 2D models, respectively. Quarter model was usually considered as an accurate model. Figure12 shows the equivalent global quarter model and the meshed solder joints. The critical solder joints, for example, solder joints under chip or component corner as shown in Fig. 12, were modeled as real shape solder joint and others were modeled as equivalent cuboid-shape solder joints with the similar effective stiffness as real shape solder joint. The submodel in the effective solder joint model is the same as that used in the quarter model as shown in Fig. 11. The purpose of considering different FE models is to investigate the accuracy of fatigue life prediction when using different FE models. Figure 13 shows the comparison of accumulated volume-averaged strain energy density per cycle of interface layer between solder and component for different models. It can be seen that 2D model gives larger plastic strain energy density than 3D models, thus means that 2D model will underestimate the fatigue life of solder joints compared to 3D models. The consistent FEA results can be obtained for quarter, octant and effective quarter models with using submodeling technique. The comparison between fatigue life prediction and experimental results also verified the accuracy of submodeling application for PBGA assembly [14].

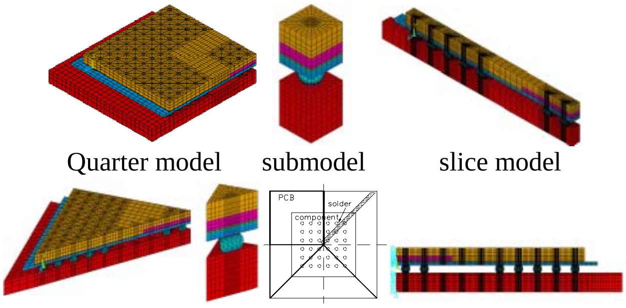


Fig. 11 2D and 3D FE models for PBGA

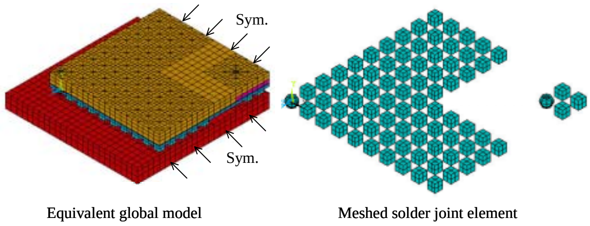


Fig. 12 Effective quarter global FE model

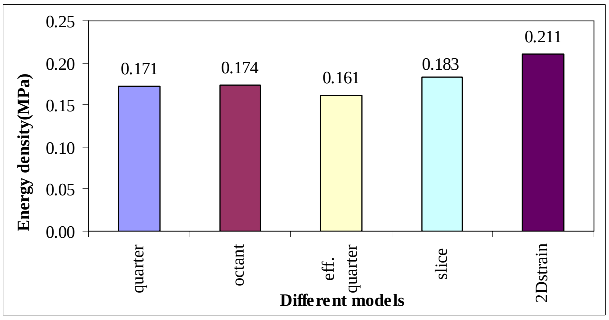


Fig. 13 Plastic work density for different models

### *3.2 FCOB Assembly Subjected to Vibration*

FCOB assembly as shown in Fig. 14 was selected for vibration test and simulation. Six larger chip modules and six smaller chip modules were mounted on the FR-4 PCB. The clamped-clamped along the longer edges boundary condition was used. The details for vibration test and analysis can be referred to publications [15-16]. The modal analysis was conducted to determine the natural frequency. First, the bare PCB was simulated for modal analysis. In addition, each chip will be modeled in order to obtain more accurate results. Because many chip and I/O connectors for FCOB assembly, it is difficult to use the traditional fine 3D model for board level simulation. So, the global-local beam (GLB) model was used in which the IC chips and PCB were modeled as shell and the solder joints were modeled as effective two-node beam elements. The natural frequency of the bare PCB with the clamped-clamped boundary condition can be obtained by [17]:

$$f_n = \frac{3.55}{a^2}\sqrt{\frac{D}{\rho}} \tag{5}$$

where $D = \frac{Eh^3}{12(1-\mu^2)}$, $\rho = \frac{Mass}{Area}$, $E$ is the Young's modulus, $\mu$ is the Poisson's ratio, $h$ is the PCB thickness, $a$ is the PCB edge thelength of, $\rho$ is the area density.

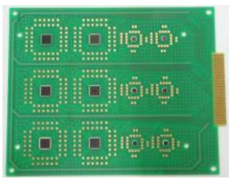


Fig. 14 FCOB assembly layout

Table 2 lists the first natural frequencies obtained from different methods. The frequency obtained from bare PCB has a good agreement with theoretical result. The result of modal analysis using global-local beam (GLB) model has a good agreement with test result.

Table 2. Comparison of the natural frequencies

| Model | Bare PCB | vs. Eq. (5) | GLB model | vs. Test |
|---|---|---|---|---|
| Frequency | 208.6 | 209.4 | 201.9 | 194.1 |

### *3.3 VQFN Assembly Subjected to Bend*

Figure 15 shows the VQFN specimen and schematics of four-point bend test. It is clear that packages between loading span subjected to similar moment load, which increases the sample size in one single four-point bend test. The details for bend test result and analysis can be referred to publications [18-20]. This paper just focuses on the FEA simulation methodology.

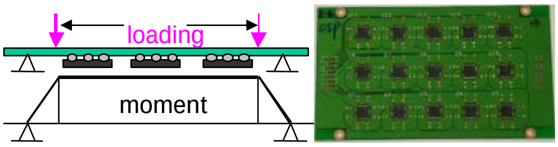



Fig. 15 VQFN and schematics of four-point bend test

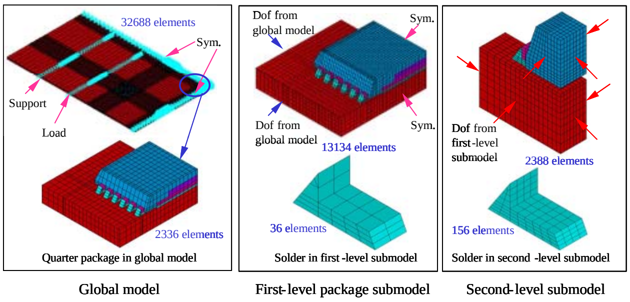



Fig. 16 Two-level submodeling method

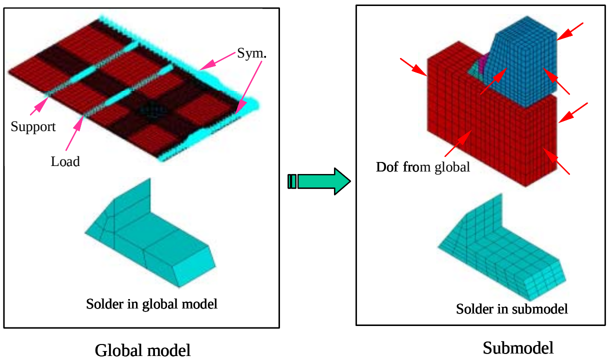



Fig. 17 One-level submodeling method

Two submodeling methods, including two-level submodeling and one-level submodeling, were compared to investigate meshing effect on FEA results. For two-level submodeling method as shown in Fig. 16, the board-level global quarter model was solved firstly. Then DOF results were transferred and interpolated to the cut boundary of the first package-level submodel. Finally, the DOF results from the first-level submodel were transferred and interpolated to the cut boundary of the second-level submodel. The package-level FEA model is referred as a transitional model, which connects the board-level global model to solder joint-level submodel. It was expected that the two-level submodeling method leads to more accurate results but complicated procedure is needed due to twice DOF interpolations. For one-level or traditional submodeling as shown in Fig. 17, the global model and submodel are the same as board-level global model and solder joint-level submodel used in two-level submodeling method, respectively. The DOF results from global model were transferred to the final submodel directly without using package-level transitional model. However, it maybe gives arise to more error because DOF results were interpolated and transferred from much coarse mesh in the board-level global model to finer mesh in the submodel. In this study, these two submodeling methods were performed for comparison.

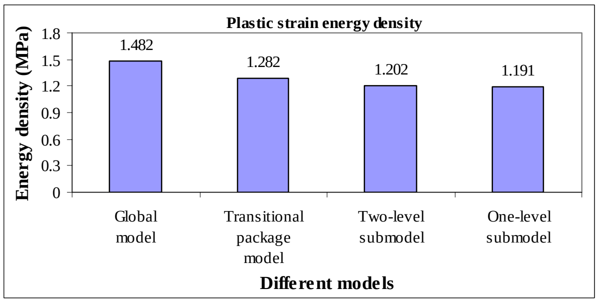



Fig. 18 Solder joint plastic strain energy density for different models

For convenience, the monotonic four-point bending load was used in this validation study. The displacement range from zero to 6mm was added on loading position of global model in 1 second using 6 load steps in FEA simulation. Figure 18 shows the volume-averaged plastic strain energy density at the maximum deflection position for different models considering whole solder joint as an averaging volume. It can be seen that one-level submodel and two-level submodel result in almost the same result. Therefore, one-level submodeling method is effective and sufficient for bending FEA modeling and simulation of VQFN assembly. The FEA simulation accuracy was verified through strain comparison between FEA modeling results and experimental results in references [19-20]. The bending fatigue model for VQFN solder joint has been proposed by combining FEA simulation results and bending reliability testing data [19-20].

## 4. Conclusions

The global-local modeling method containing submodeling and GLB modeling technique was developed and calibration study was also conducted. Reasonable submodel with hybrid solid, including solder joint, PCB and component, can give more accurate result than submodel with only solder joint. Solid-beam-solid GLB model can lead to more accurate results than shell-beam-shell GLB model. The modified shell model gives less error than non-modified shell elements. GLB model is reasonable for small deformation problem and modal analysis because only elastic deformation behavior is considered for solder joint in the deformation analysis of entire structure. Based on GLB modeling theory, the ball shape solder joint can be replaced by the effective solder joint with cubic cross-section assuming that they have similar stiffness. The effective solder joint model can lead to accurate result with solder joint plastic deformation consideration.

Case studies further validate the feasibility and accuracy of global-local modeling technique used in electronic packaging reliability assessment. Three cases are considered, including PBGA assembly subjected to thermal cycling, FCOB assembly subjected to vibration and VQFN assembly subjected to bend.

**References**


1. Che, F.X., and Pang, H.L.J., "Thermal Fatigue Reliability Analysis for PBGA with Sn-3.8Ag-0.7Cu Solder Joints", *Proc 6th Electronics Packaging Technology Conf*, Singapore, Dec. 8-10, 2004, pp. 787-792.
2. Pang, H.L.J., Chong, D.Y.R., and Low, T.H, "Thermal Cycling Analysis of Flip-Chip Solder Joint Reliability", *IEEE Transaction on Components and Packaging Technologies*, Vol. 24, No.4 (2001), pp. 705-712.
3. Pang, H.L.J., and Chong, Y.R., "Flip Chip on Board Solder Joint Reliability Analysis Using 2-D and 3-D FEA Models", *IEEE Transaction on Advanced Packaging*, Vol.24, No.4 (2001), pp. 499-506.
4. Akay, H.U., Liu, Y., and Rassaian, M., "Simplification of Finite Element Models for Thermal Fatigue Life Prediction of PBGA Packages", *Journal of Electronic Packaging*, Vol.125 (2003), pp. 347-353.
5. Pang, H.L.J., Seetoh, C.W., and Wang, Z.P., "CBGA Solder Joint Reliability Evaluation Based on Elastic-Plastic-Creep Analysis", *Journal of Electronic Packaging*, Vol. 122 (2000), pp. 255-261.
6. Zhu, J., Quander, S., and Reinikainen, T., "Global/Local Modeling for PWB Mechanical Loading", *Proc 51th Electronic Components and Technology Conf*, Orlando, May 29-June 1 2001, pp. 1164-1169.
7. Gustafsson, G., Guven, I., Kradinov, V., *et al.*, "Finite Element Modeling of BGA Packages for Life Prediction", *Proc 50th Electronic Components and Technology Conf*, Las Vegas, May 21-24, 2000, pp. 1059-1063.
8. Pang, H.L.J., Low, T.H., Xiong, B.S., and Che, F.X., "Design For Reliability (DFR) Methodology For Electronic Packaging Assemblies", *Proc 5th Electronics Packaging Technology Conf*, Singapore, Dec. 10-12, 2003, pp. 470-478.
9. Yang, Q. J., Lim, G.H., Pang, H.L.J., et al., "Vibration Reliability Analysis of a PBGA Assembly under Foundation Excitations", *EEP-Vol. 26-1, Advances in Electronic Packaging, Vol. 1, ASME International Electronic Conf*, 1999, pp. 705-711.
10. Chong, Y.R., Che, F.X., Xu, L.H., Toh, H.J., Pang, J.H.L., Xiong, B.S., and Lim, B.K., "Performance Assessment on Board-level Drop Reliability for Chip Scale Packages (Fine-pitch BGA)", *Proc 56th Electronic Components and Technology Conf*, San Diego, California, May 30-June 2, 2006, pp. 356-363.
11. Che, F. X., Pang, H.L.J., Zhu, W.H., Sun, W., and Sun, Y. S., "Modeling Constitutive Model Effect on Reliability of Lead-Free Solder Joints", *Proc Internatioanl Conf Electronic Packaging Technology,* Shanghai, Aug. 27-29, 2006, pp. 155-160.
12. ANSYS Version 7.0 Manual, 2002, *Ansys Inc*.
13. Petyt, M., Introduction to Finite Element Vibration Analysis, *Cambridge University Press*, 1990.
14. Che, F.X., Pang, H.L.J., Xiong, B.C., Xu, L.H., and Low, T.H., "Lead Free Solder Joint Reliability Characterization for PBGA, PQFP and TSSOP Assemblies", *Proc 55th Electronic Components and Technology Conf*, Florida, May 31-June 3, 2005, pp. 916-921.
15. Pang, H.L.J., Che, F.X., and. Low, T.H., "Vibration Fatigue Analysis For FCOB Solder Joints", *Proc 54th Electronic Components and Technology Conf*, Las Vegas, Nevada, June 1-4, 2004, pp. 1055-1061.
16. Che, F.X., Pang, H.L.J., Wong, F.L., Lim, G.H., and Low, T.H., "Vibration Fatigue Test and Analysis for Flip Chip Solder Joints", *Proc 5th Electronics Packaging Technology Conf*, Singapore, December 10-12, 2003, pp. 107-113.
17. Steinberg, Dave S., Vibration Analysis for Electronic Equipment, John Wiley & Sons (New York, 1998).
18. Che, F.X., and Pang, H.L.J., "Bend Fatigue Reliability Test and Analysis for Pb-free Solder Joint", *Proc 7th Electronics Packaging Technology Conf*, Singapore,, December 2005, pp. 868-872.
19. Pang, H.L.J., and Che, F.X., "Isothermal Cyclic Bend Fatigue Test Method For Lead Free Solder Joints", *Proc 2006 Inter Society Conference on Thermal Phenomena,* San Diego, California, May 30-June 2, 2006, pp. 1011-1017.
20. Che, F.X., and Pang, H.L.J., "Modeling Board-Level Four-Point Bend Fatigue and Impact Drop Tests", *Proc of* 56*th Electronic Components and Technology Conf*, San Diego, California, May 30-June 2, 2006, pp. 443-448.