A novel coupling simulation scheme for elastoplastic problems of multiscale structures in electronic packaging

1st Chong Ma
Institute of Electronics Packaging Technology and Reliability
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
MC07060619@163.com

2nd Fei Qin
Institute of Electronics Packaging Technology and Reliability
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
qfei@bjut.edu.cn

3rd Yanpeng Gong*
Institute of Electronics Packaging Technology and Reliability
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
yanpeng.gong@bjut.edu.cn

4th Chuantao Hou
Science and Technology on Reliability and Environment Engineering Laboratory
Beijing Institute of Structure and Environment Engineering
Beijing, China
houcht@qq.com

5th Hao Cheng
Science and Technology on Reliability and Environment Engineering Laboratory
Beijing Institute of Structure and Environment Engineering
Beijing, China
chenghao613@126.com

6th Qi He
Institute of Electronics Packaging Technology and Reliability
Faculty of Materials and Manufacturing
Beijing University of Technology
Beijing, China
heqi@emails.bjut.edu.cn

2022 23rd International Conference on Electronic Packaging Technology (ICEPT) | 978-1-6654-9905-7/22/$31.00 ©2022 IEEE | DOI: 10.1109/ICEPT56209.2022.9873417

***Abstract* —With the increasing of packaging density and functional diversity, electronic packaging structures with multiscale features have brought huge demand. To analyze elastoplastic problems of multiscale structures in electronic packaging, this paper presents an efficient coupling scheme of Abaqus and a self-written boundary element method (BEM) code. In the numerical analysis, based on the geometric features of the multiscale structures, the whole domain is divided into two domains: FE domain and BE domain. The Abaqus is used in the FE domain where non-linear or nonhomogeneous behavior is expected, whereas the linear elastic domain or large-scale domain is solved by the BEM code. The BE part in the whole FE-BE model is defined as a super element (user-defined element in Abaqus) of the FE, and its effective stiffness and effective nodal forces at the interfacial boundary are evaluated by the BEM code and assembled by the user subroutine (UEL) into the FE system. Users not only benefit from the powerful functions of Abaqus, but also can use BEM as a complement to improve solution accuracy. The numerical example shows that the stress results obtained by the scheme are in a great agreement with the results obtained by a detailed finite element model.**

Keywords—Boundary element method, Abaqus, coupling method, multiscale structures, electronic packaging

I. INTRODUCTION

Electronic packaging is a process of converting chips into reliable electronic devices in electronic manufacturing industry chain. A typical packaging structure assembly consists of various materials with discrepant mechanical and thermal properties. Geometric discontinuity feature usually exists in packaging structures because of through vias, free edges, and multiple interfaces [1]. However, when different materials are integrated to form an electronic packaging structure, the coefficient of thermal expansion mismatch will induce large thermal stresses during processing or in operation, which makes devices especially prone to thermomechanical fatigue failures. With the increasing of packaging density and functional diversity, such as 3D electronic packaging, heterogeneous integration technology, the reliability problems of electronic packaging structures with multiphysics and multiscale features are becoming one of the key requirements for the design.

Due to the characteristics of easy implementation, saving resources, no environmental restriction, the design of electronic packaging structure based on numerical method is becoming the mainstream scheme. Currently, there are many commercial programs available for reliability analysis, such as Abaqus, Ansys, Comsol, etc. In [2], Zhang and Li presented a general method to predict the thermal stress and the temperature of TSV arrays based on coupled-field finite element method (FEM). The authors proved the accuracy of the research work by the 3D finite element model and experimental measurements. In [3], Baek et al. presented a simulation method based on Representative Volume Element (RVE) to investigate the factors affecting wafer warpage, and found that Young's modulus and thermal expansion coefficient are main parameters affecting wafer warpage. In [4], An et al. optimized the vibration lifetime estimation of PBGA assemblies that is soldered with Sn37Pb under random vibration loading on the basis of vibration failure tests, finite element analysis (FEA) and Steinberg's empirical formulation. But nonlinear problem was not considered in their finite element models. In [5], Omar Ahmed et al. created plane strain FEA models to study the factors causing substrate cracking reliability problems in through glass via, and the study provides a new point for improving the reliability of TGV.

From the above study we can find that the finite element analysis is a mainstream method to study the reliability problems of electronic packaging structures. Actually, due to the multiscale features of electronic packaging structures, the numerical model built by FEM should be discretized into large number of elements to ensure the rationality of numerical solution. Especially, for some complex models with multiscale features, the computational time and cost might be unacceptable. Since the advantages of reduced dimensionality, boundary discretization and high precise analysis, the boundary element method is then extended and used to analyze electronic packaging reliability problems. In [6], Yu et al. used the isogeometric BEM to carry out the heat transfer problems of packaging structures, and several examples are used to verify the accuracy of the method. In [7], Vallepuga-Espinosa et al. applied BEM to study the thermoelastic contact problem of electronic packaging with low heat generation. Although the number of elements can be largely reduced by BEM in the analysis, the BEM is not a preferable approach for the nonlinear and nonhomogeneous problems.

Since the respective advantages of boundary element method and finite element method, this paper presents an efficient coupling scheme of commercial software Abaqus and a BEM code for elastoplastic problems in electronic packaging. In the implementation of the current coupling scheme, the whole domain should be split into FE domain and BE domain. The Abaqus is used in the sub-domain where nonlinear or nonhomogeneous behavior is expected, whereas the linear behavior or unimportant sub-domain is solved by the BEM code. The interface (i.e. the equivalent BE sub-domain) between the FE domain and BE domain is treated as "finite super elements". The effective stiffness and effective nodal forces of these super elements can be evaluated by the BEM code. Then, the obtained effective quantities can be assembled to the global FE formulations. Through this way, the FE sub-domain can be solved by Abaqus.

II. Basic Theoretical Preparation

In this section, we will briefly derive the theoretical basis of the coupled scheme. More details about the coupling theory can be found in [8].

To couple BEM to FEM, the boundary tractions $\{t_c\}$ in BEM should be converted into equivalent nodal forces $\{F_c\}$ in FEM. Consider the model in Fig.1, consisting of one BE domain and one FE domain. For the BE domain, the relationship between displacements $\{u_c\}$ and tractions $\{t_c\}$ at the interface can be expressed as [8]

$$\{t_c\} = \{t_{c0}\} + [K_{\mathrm{BE}}]\{u_c\} \tag{1}$$

where $\{t_{c0}\}$ is a vector containing tractions, and $\{K_{\mathrm{BE}}\}$ is the nominal stiffness matrix of the BE domain.

We apply an arbitrary virtual displacement δu_x^e in x-direction at a random node along the interface element e. Then, the work done by nodal forces F_x^e can be expressed as

$$\delta W_x^{(e)} = \sum_{i=1}^{n} (F_{ix}^e \delta u_{ix}^e) \tag{2}$$

where n is the number of nodes for anyone element.

Due to the principle of conservation of energy, the work done by the surface tractions (t_x) should be equal to that done by the equivalent nodal forces (F_x), which gives

$$\delta W_x^{(e)} = \int_\Gamma t_x \delta u_x d\Gamma \tag{3}$$

where Γ is the field of integration (shown in Fig. 1), the tractions t_x and virtual displacements δu_x can be easily interpolated as

$$\delta u_x = \sum_{i=1}^{n} N_i \delta u_{ix}^e \tag{4}$$

$$t_x = \sum_{j=1}^{n} N_j t_{jx}^e \tag{5}$$

where N_i and N_j are the shape function.

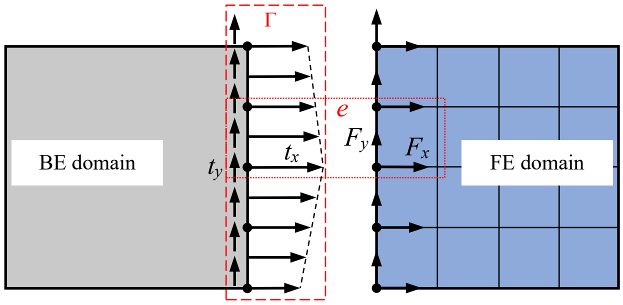

Fig. 1. Interface between FE and BE sub-domains showing interface forces

From (2) to (5), we can obtain that

$$F_{ix} = \sum_{j=1}^{n} t_{ix} \int_\Gamma N_i N_j d\Gamma \tag{6}$$

Similarly, we can obtain

$$F_{iy} = \sum_{j=1}^{n} t_{iy} \int_\Gamma N_i N_j d\Gamma \tag{7}$$

The equivalent nodal forces can be calculated by the following formulation

$$\{F_c\} = [M]\{t_c\} \tag{8}$$

where $\{F_c\}$ is nodal forces at the interface, and matrix $[M]$ which is assembled by $[M^e]$ denotes transformation matrix for the whole interface. The matrix $[M^e]$ can be obtained by the following formulation

$$[M^e] = \sum_{j=1}^{n} \int_\Gamma N_i N_j d\Gamma \tag{9}$$

where n is the number of nodes for one element.

Then, (1) can be expressed in terms of equivalent nodal forces by pre-multiplying with the matrix $[M]$

$$\{F_c\} = [M]\{t_c\} = [M]\{t_{c0}\} + [M][K_{\mathrm{BE}}]\{u_c\} \tag{10}$$

where $[M][K_{\mathrm{BE}}]$ is the effective stiffness matrix in the finite element part.

In the implementation of the coupling method, the BE domain is treated as a finite super element and its stiffness matrix $[K_{\mathrm{BE}}]$ can be evaluated by the above description.

III. Implementation Of Coupling Scheme

To make the current coupling scheme more acceptable by engineers, we introduce an automatic procedure to implement the coupling scheme. The coupling scheme will be implemented through Abaqus. Fig. 2 shows the technique that combines the BEM code and the commercial software Abaqus. The implementation process is as follows:

1) The model is divided into FE and BE sub-domains according to its geometric features or materials properties.

2) The FE part and the equivalent BE part should be created by Abaqus. The FE part is created by a standard operation in Abaqus/CAE. The equivalent BE part is just one line along the interface. To improve the accuracy of numerical results, the number of interfacial nodal points for the two parts is equal. The FE and BE parts are connected by the command 'Tie' of Abaqus.

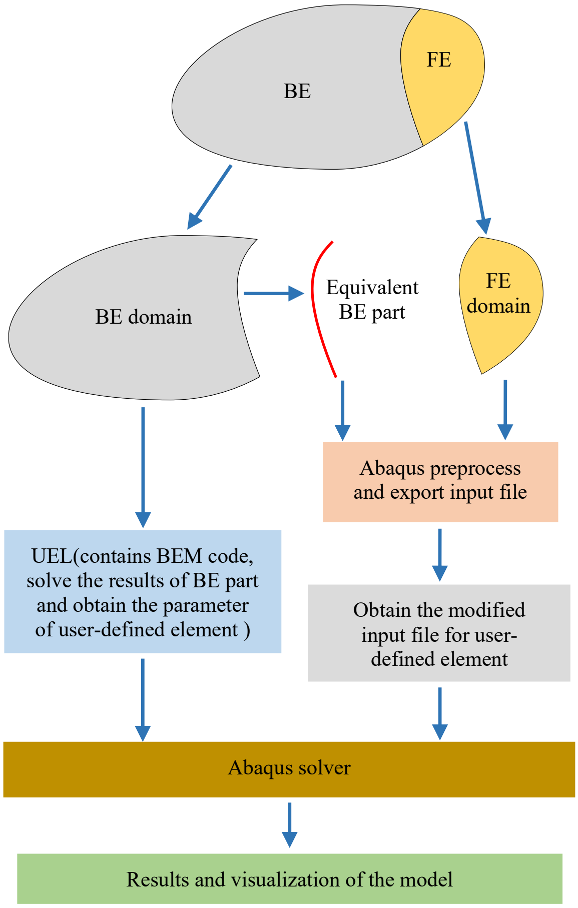

Fig. 2. The flowchart of coupling scheme

3) After the two parts being created, the modeling procedure (including Property, Assembly, Step, Load and Mesh) for the coupling scheme is the same as a pure FE model.

4) The inp file produced by Abaqus need to be modified to define the equivalent BE part as user-defined element [9].

5) Then, the modified input file and the subroutine UEL (including the self-written BEM code) are called to perform the coupling scheme. All BE data (i.e. nodes, boundary elements, materials of each part, interfaces, loads, boundary conditions) will be provided in the UEL. The effective nodal forces and stiffness matrix of the user-defined element are obtained by the UEL.

6) Finally, post-processing functions can be used to view the results.

IV. NUMERICAL EXAMPLE

In the following, a representative numerical example is used to demonstrate the performance of the current coupling scheme.

Fig. 3 shows a cross-section of through silicon via (TSV) structure. The Cu in TSV is produced by electroplating process and the Cu over-burden will be inevitable on the surface. In the production process, the over-burden needs to be removed by chemical mechanical polishing (CMP) [10]. In this example, a simplified plane strain model (shown in Fig. 3) is developed to study the stress and displacement when engineers remove the over-burden (Cu). The geometric parameters of the model are given as a = 630 μm, b = 250 μm, d = 10 μm, h = 100 μm and δ = 6 μm. The spacing between each blind via is 150 μm. As shown in Fig. 3, the bottom of Si part is fixed and the surface of over-burden is subjected to shear traction t = 0.25 N/m. The material properties are listed in TABLE I.

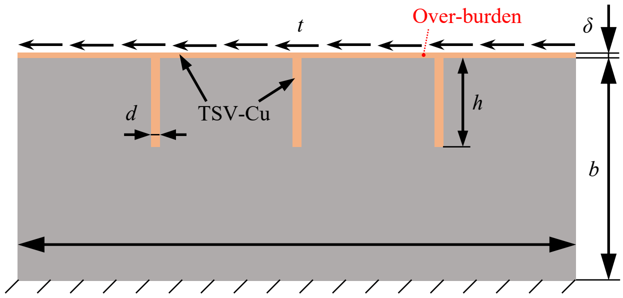

Fig. 3. Cross-section of TSV

TABLE I. MATERIAL PARAMETERS OF TSV

Part	Properties		
	Young's Modulus (MPa)	*Poisson's Ratio*	*Yield Strength* (MPa)
TSV-Cu	155000	0.3	47.91
Si	140000	0.25	--

As introduced above, the numerical model is divided into FE domain and BE domain. Due to the influence area of the shear traction on the model is near the over-burden, and the boundary condition has limited influence on the TSV-Cu part. Therefore, in the analysis, the finite element analysis is used to study the TSV-Cu part and the BEM is used to analyze the Si part. Fig. 4 shows the details of meshing used in the coupling scheme, from which we can find that quadrilateral element (CPE4R) is used in the TSV-Cu part and boundary element (user-defined element) is used to discretize the boundary of the Si part.

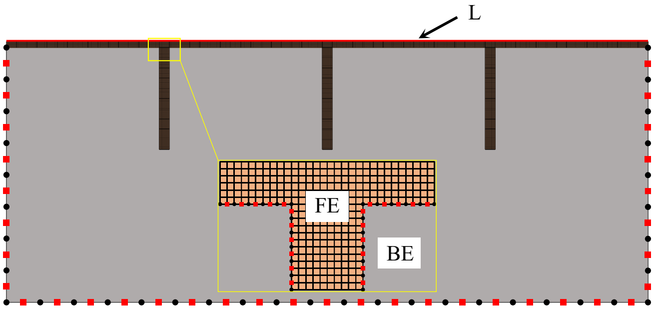

Fig. 4. Meshes used in the coupling scheme

Fig. 5 shows the contour plot of the von Mises stress in the TSV-Cu part obtained by the current coupling scheme. For comparison, the contour plot of finite element analysis with very refined mesh is provided in Fig. 6.

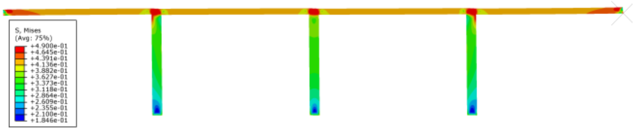

Fig. 5. Stress obtained by the coupling scheme

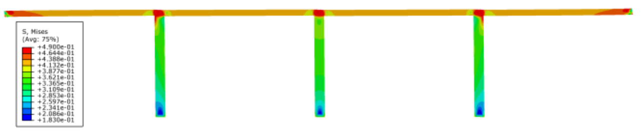

Fig. 6. Stress obtained by FEA

To further verify the accuracy of the coupling scheme, we plot in Fig. 7 the von Mises stress at points along the contour L (shown in Fig. 4). From Fig. 7, we can find that the stress results obtained by the current coupling scheme have a great agreement with that obtained by FEA.

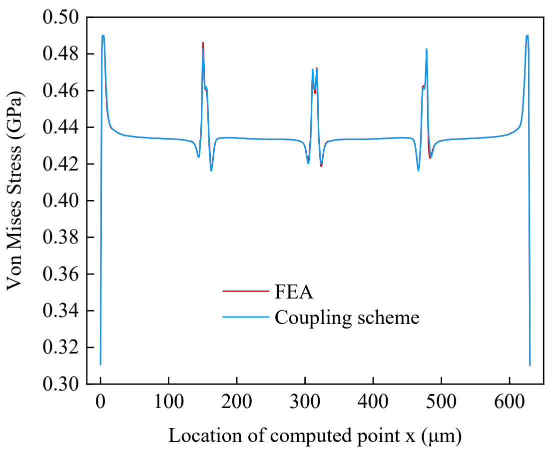

Fig. 7. The von Mises stress at points along the contour L

V. CONCLUSIONS

This paper presents a convenient and efficient coupling scheme of the commercial software Abaqus and a self-written BEM code. In the current coupling scheme, based on the geometric features of the multiscale structures, the whole model is divided into the BE sub-domain and the FE sub-domain. The self-written BEM code will be implemented using the built-in UEL subroutine in Abaqus to get the stiffness and tractions for the user-defined element. Then, the FE domain can be solved by using the obtained effective stiffness and forces. The numerical example shows that very accurate results can be obtained by the coupling scheme.

ACKNOWLEDGMENT

The research was supported by the National Natural Science Foundation of China (No. 12002009), the General Program of Science and Technology Development Project of Beijing Municipal Education Commission (No. KM202110005032).

REFERENCES

[1] S. Liu, Y. Liu, Modeling and simulation for microelectronic packaging assembly: manufacturing, reliability and testing, Hoboken: Wiley, 2011.

[2] C. Zhang, L. Li, "Characterization and design of through-silicon via arrays in three-dimensional ICs based on thermomechanical modeling", IEEE TRANSACTIONS ON ELECTRON DEVICES, vol.8, pp. 279-287, 2011.

[3] J. W. Baek, W. S. Yang, M. J. Hur, "Represnetative volume element analysis for wafer- level warpage using finite element methods", Materials Science in Semiconductor Processing, vol.91, pp. 392-398, 2019.

[4] T. An, F. Qin, B. Zhou, P. Chen, Y. Dai, "Vibration lifetime estimation of PBGA solder joints using Steinberg model", Microelectronics Reliability, vol.102, 113474, 2019.

[5] O. Ahmed, C. Okoro, S. Pollard, T. Jiang, "The effect of materials and design on the reliability of through-glass vias for 2.5D integrated circuits: a numerical study", Multidiscipline Modeling in Materials and Structures, vol.17, pp. 451-464, 2021.

[6] H. Yu, Y. Guo, Y. Gong, F. Qin, "Thermal analysis of electronic packaging structure using isogeometric boundary element method", Engineering Analysis with Boundary Elements, vol.128, pp. 195-202, 2021.

[7] J. Vallepuga-Espinosa, I. Ubero- Martínez, L. Rodríguez-Tembleque, J. Cifuentes-Rodríguez, "A boundary element procedure to analyze the thermomechanical contact problem in 3D microelectronic packaging", Engineering Analysis with Boundary Elements, vol.115, pp. 28-39, 2020.

[8] G. Beer, I. Smith, C. Duenser, The boundary element method with programming: for engineers and scientists, Springer: Vienna, 2008.

[9] S. D., Abaqus 2016 analysis user's manual, Abaqus 2016 Documentation, 2016.

[10] S. C. Hong, G. L. Wang, W. J. Kim, J. H. Kim, J. P. Jung, "Reduction of defects in TSV filled with Cu by high-speed 3-step PPR for 3D Si chip stacking", Microelectronics Reliability, vol.51, pp. 2228-2235, 2011.