ICCES, 2025, vol.33, no.4

Tech Science Press

DOI: 10.32604/icces.2025.011409

PROCEEDINGS

Stress Intensity Factor Analysis for Non-Homogeneous Materials Based on Secondary Development of ABAQUS

Fengnan Guo[1,*], Yiming Li[1], Hua Zhang[1] and Jianwei Cui[2]

[1]School of Aeronautics and Astronautics, Dalian University of Technology, Dalian, 116024, China

[2]The 41st Institute of Sixth Academy of CASIC, Huhhot, 010010, China

[*]Corresponding Author: Fengnan Guo. Email: guofn@dlut.edu.cn

ABSTRACT

The stress intensity factor (SIF) is one of the most crucial parameters in fracture mechanics, as it can effectively characterize the state of the crack and determine its propagation behavior. The methods for evaluating the stress intensity factor mainly include the J-integral method, interaction integral method, and displacement extrapolation method [1,2]. However, the conventional J-integral and interaction integral methods involve derivative terms of material parameters, which cause a great difficulty in applying these methods to deal with non-homogeneous materials containing material interfaces. In order to overcome this difficulty, an improved interaction integral method has been proposed [3]. This approach introduces a suitably designed auxiliary field to eliminate the derivative terms of material parameters in the interaction integral, which enables the direct extraction of the stress intensity factor at the crack tip without considering the material interfaces. Based on this form of the interaction integral, this paper utilizes the secondary development of ABAQUS to extract the simulation results from the specified analysis step. By programming the post-processing results, the study achieves the rapid computation of the novel interaction integral [4]. To verify the accuracy of the program, examples of homogeneous materials containing cracks were selected for the calculation of crack parameters. The results demonstrate that the program's calculations are in good agreement with those obtained from theory and simulation. Finally, examples with complex interfaces in two and three dimensions were selected to calculate fracture parameters. Compared with simulation results, the self-developed numerical procedure in this study demonstrates robust domain independence when calculating fracture parameters in materials with complex interfaces. This new method avoids intermediate data processing, improves the efficiency of parameter extraction, and extends the application scope of ABAQUS for calculating fracture parameters of non-homogeneous materials, especially for those integral domains intersecting other interfaces.

KEYWORDS

Stress intensity factor; interaction integral method; ABAQUS secondary development

In this paper, examples of non-homogeneous materials containing inclusions were chosen, and the results were compared with the outputs from ABAQUS. As shown in Fig. 1 and Fig. 2, a certain number of inclusions are randomly distributed within the matrix, with an edge crack on one side of the matrix and both sides are subjected to unidirectional tensile loads. The stress intensity factors at the crack tip are calculated using the self-developed program presented in this paper, and the results are compared with ABAQUS outputs. The comparisons of the computational results from the two methods are shown in Figure 3 and Figure 4. The computational results show that when the integration region is small and does not include the material interfaces, the results from both methods are almost identical. However, when the integration regions cross the inclusion, due to the inability to deal with derivative terms of material parameters at the interfaces, the stress intensity factors calculated by ABAQUS will undergo significant changes. As a result, the computational results are different in different integration regions, making it impossible to accurately determine the crack situation in this case. In contrast, the interaction integral in the self-developed program eliminates the derivative terms of material parameters. Even when the integration regions cross the inclusion, the calculated stress intensity factors still maintain the domain independence. The method presented in this paper enhances the computational efficiency of the stress intensity factors and broadens the application scope of ABAQUS.

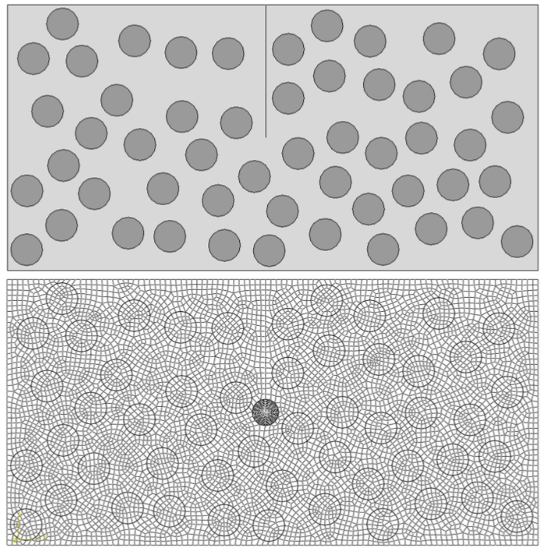

Figure 1: Numerical example of 2D non-homogeneous materials with inclusions and its mesh.

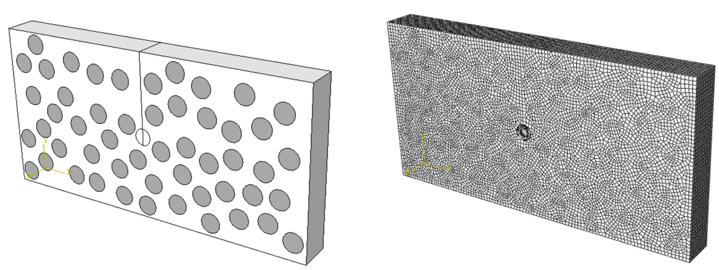

Figure 2: Numerical example of 3D non-homogeneous materials with inclusions and its mesh.

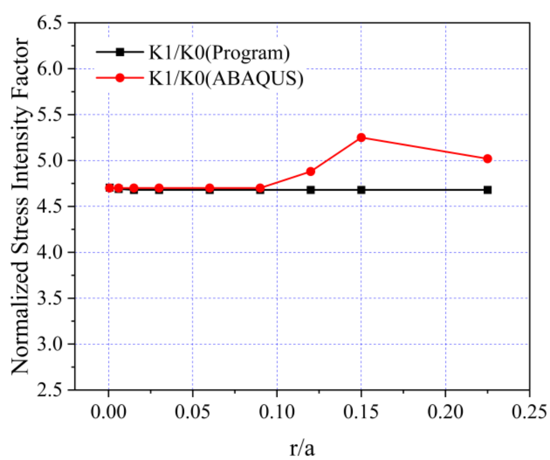

Figure 3: Comparison of computational results between the self-developed program and ABAQUS for two-dimensional examples.

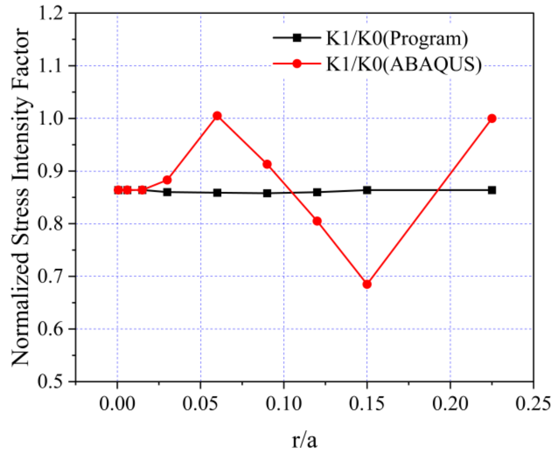

Figure 4: Comparison of computational results between the self-developed program and ABAQUS for three-dimensional examples.

Funding Statement: This work is supported by the National Natural Science Foundation of China (Grant No. 12372075).

Conflicts of Interest: The authors declare that they have no conflicts of interest to report regarding the present study.

References

1. Rice, J. R., Sih, G. C. (1965). Plane problems of cracks in dissimilar media. *Journal of Applied Mechanics, 32,* 418–423.
2. Stern, M., Becker, E. B., Dunham, R. S. (1976). A contour integral computation of mixed-mode stress intensity factors. *International Journal of Fracture, 12 (3),* 359–368.
3. Yu, H. J., Wu, L. Z., Guo, L. C., et al. (2009). Investigation of mixed-mode stress intensity factors for nonhomogeneous materials using an interaction integral method. *International Journal of Solids and Structures, 46(20),* 3710–3724.
4. Li, Y. M., Wang, K., Zhang, H., Guo, F. N. (2024). Calculation of fracture parameters of solid rocket motor grain based on the secondary development of ABAQUS. *Proceedings of the 8th China Joint Conference of Aerospace Propulsion,* 1031–1037. Chengdu, China.