ICCES, 2023, vol. 26, no.3

Tech Science Press

DOI: 10.32604/icces.2023.09023

PROCEEDINGS

A Peridynamics-Based Finite Element Method (PeriFEM) and Its Implementation in Commercial FEM Software for Brittle Fractures

Fei Han[1,*], Zhibin Li[1], Jianyu Zhang[1], Zhiying Liu[1], Chen Yao[1] and Wenping Han[1]

[1] State Key Laboratory of Structural Analysis for Industrial Equipment, Department of Engineering Mechanics, Dalian University of Technology, Dalian, 116023, China

[*]Corresponding Author: Fei Han. Email: hanfei@dlut.edu.cn

ABSTRACT

The classical finite element method has been successfully applied to many engineering problems but not to cases with space discontinuity. A peridynamics-based finite element method (PeriFEM) is presented according to the principle of minimum potential energy, which enables discontinuity. First, the integral domain of peridynamics is reconstructed, and a new type of element called peridynamic element (PE) is defined. Although PEs are generated by the continuous elements (CEs) of classical FEM, they do not affect each other. Then, spatial discretization is performed based on PEs and CEs, and the linear equations about nodal displacement are established according to the principle of minimum potential energy. Besides, cracks are characterized by the degradation of the mechanical properties of PEs. Consequently, PeriFEM is a reformulation of the traditional FEM for solving peridynamic equations numerically. It considers the non-local features of peridynamics yet possesses the same computational framework as the traditional FEM. As a result, this implementation benefits from the consistent computational frameworks of both PeriFEM and traditional FEM. After that, we propose the first unified implementation strategy for peridynamics in commercial FEM software packages based on their application programming interface using the PeriFEM. Using ANSYS and ABAQUS as examples, we present the numerical results and implementation details of PeriFEM in commercial FEM software. The codes integrated into ANSYS and ABAQUS are both verified through benchmark examples, and the computational convergence and costs are compared. The results show that ABAQUS is more efficient for some specific examples, whereas the convergence criterion adopted in ANSYS is more robust. Finally, 3D examples are presented to demonstrate the ability of the proposed approach to deal with complex engineering problems.

Funding Statement: The authors received no specific funding for this study.

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