Finite Elements in Analysis and Design 126 (2017) 26–38

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

journal homepage: www.elsevier.com/locate/finel

Accurate finite element simulation of stresses for stationary dynamic cracks under impact loading

A. Idesman[a,*], A. Bhuiyan[a], J.R. Foley[b]

[a] Department of Mechanical Engineering, Texas Tech University, Lubbock, TX 79409-1021, USA
[b] Air Force Research Laboratory, Munitions Directorate, Eglin Air Force Base, FL 32542, USA

ARTICLE INFO

Keywords:
Cracks
Elastic waves
Finite elements
Numerical dispersion
Dynamic stress intensity factor

ABSTRACT

The numerical technique for wave propagation problems recently developed in our papers has been applied to the accurate modeling of stresses in the vicinity of crack tips and the dynamic stress intensity factor (DSIF) for stationary cracks. The numerical technique includes the linear finite elements with reduced dispersion as well as the two-stage time-integration approach that quantifies and filters spurious high-frequency oscillations. Several benchmark problems for stationary cracks at impact loadings have been solved. The accuracy of the stress calculation in the vicinity of the crack tips and the DSIF can be significantly increased by the application of the finite elements with reduced dispersion. Surprisingly, even without a special treatment of singularities at crack tips, the linear finite elements with reduced dispersion (with no crack tip enrichment functions) yield much more accurate results than the XFEM with the special crack tip enrichment functions on comparable meshes. It is also interesting to mention that for the calculation of the DSIF by the finite elements with reduced dispersion there is no necessity in the filtering stage at impact loading (the spurious oscillations in the DSIF are small and decrease with mesh refinement).

1. Introduction

Accurate simulations of stresses in the vicinity of crack tips under dynamic loading are very important for the prediction of the failure of different structural components. Due to the simplicity of its application, the finite element method (including its different modifications) is one of the popular methods in fracture mechanics. Let us describe some known approaches for the simulation of cracks under dynamic loadings; e.g., see [5,8–10,12,14,15,32,34,37,39,42,44,48,49,53,54] and many others. One of the peculiarities for such problems is the appearance of a singularity for the stress-strain state in the vicinity of the crack tip. Several different techniques have been suggested to treat this singularity such as the X-FEM method (e.g., see [5,8-10,14-16,32,34,39,40,42-44,49]), the displacement-based methods with special 'quarter-point' finite elements (e.g., see [12]), the special enrichment for the smoothed finite element (e.g., see [37]), the virtual crack closure technique (e.g., see [54]), the direct mesh refinement in the vicinity of the crack tip (e.g., see [48,53]) and other.

One of the most important factors that affects the accuracy of numerical solutions of elastodynamics problems (even without cracks) is the numerical dispersion; e.g., see [6,7,13,17,19,20,33,38,41,46,47,52,55,56]. Recently, different techniques have been developed for the reduction of the numerical dispersion error for the linear and high-order finite elements used for dynamics problems; e.g., see [1,2,6,7,13,17-20,24,27,33,38,41,46,47,50-52,55,56].

Another issue in the numerical modeling of dynamics problems under high-frequency and impact loadings is the appearance of spurious oscillations that may significantly reduce the accuracy of the numerical results. Despite the fact that many dynamic benchmark problems in fracture mechanics are considered under impact loading we have not seen in the literature the treatment of spurious high-frequency oscillations in the numerical solutions to such problems. These oscillations may significantly affect the stress distribution and the prediction of the crack behavior; e.g., see Fig. 2b.

In this paper, we will apply the accurate finite element technique with reduced dispersion to the simulation of stresses for stationary cracks under impact loading. In Sections 2.1 and 2.2 we will briefly summarize the finite element technique with reduced dispersion and the two-stage time-integration approach with the quantification and filtering of spurious oscillations recently developed in our papers [22-24,27,29-31]. The formulas for the calculation of the dynamic stress intensity factor are presented in Section 2.3. The numerical examples of the application of the accurate numerical technique to the simulation

* Corresponding author.
E-mail address: alexander.idesman@ttu.edu (A. Idesman).

http://dx.doi.org/10.1016/j.finel.2016.12.004
Received 17 August 2016; Received in revised form 8 December 2016; Accepted 12 December 2016
0168-874X/

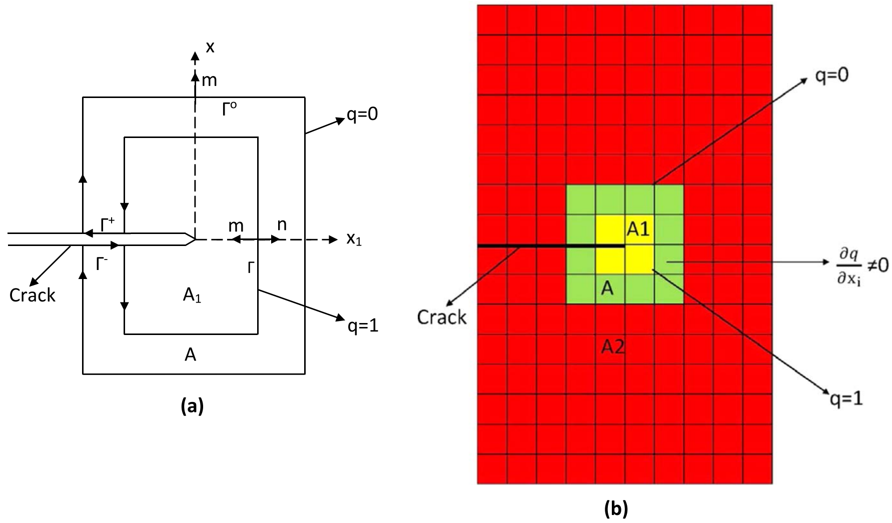

Fig. 1. A schematic diagram for the calculation of the J integral in the vicinity of the crack tip (a). A finite element mesh in the vicinity of the crack tip (b). x_1 and x_2 are the local Cartesian axes with the x_1-axis parallel to the crack face.

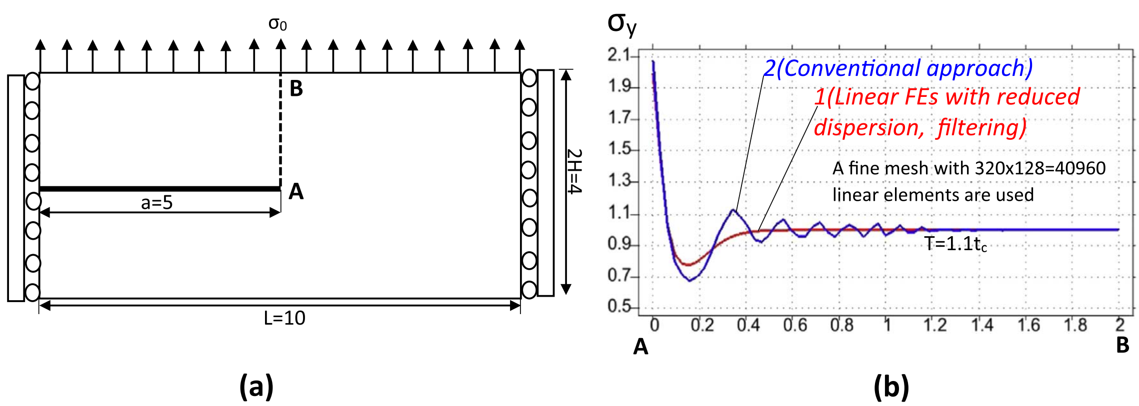

Fig. 2. Geometry of a plate with a crack loaded by a tensile stress: (a) The distribution of the σ_y stress along line AB in the vicinity of the crack tip A at time $T = 1.1t_c$. (b) Curve 1 corresponds to the numerical results obtained by the linear elements with reduced dispersion after the filtering stage. Curve 2 corresponds to the numerical results obtained by the conventional finite elements without the filtering stage. A fine uniform mesh with $320 \times 128 = 40,960$ linear finite elements is used.

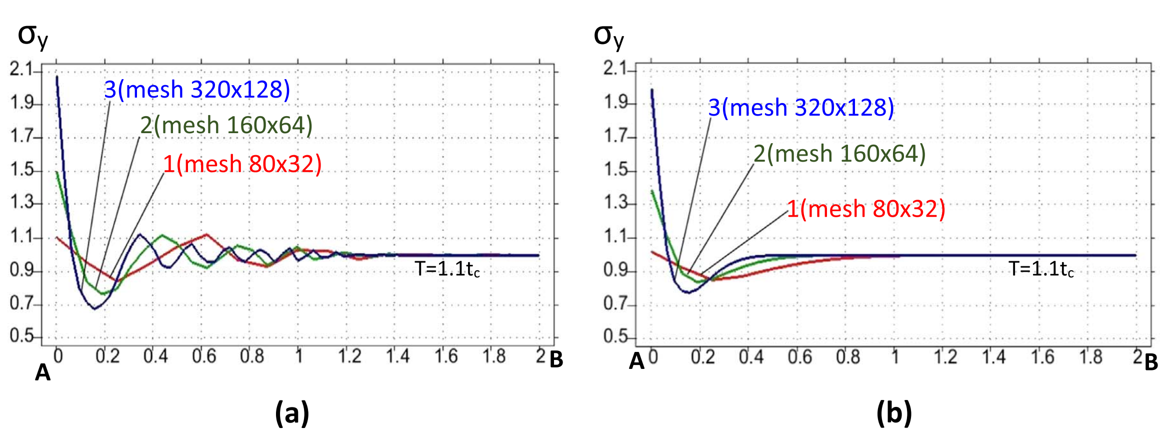

Fig. 3. The distribution of the σ_y stress along line AB in the vicinity of the crack tip A at time $T = 1.1t_c$ obtained by the conventional finite elements without the filtering stage (a) and by the linear elements with reduced dispersion after the filtering stage (b). Curves 1, 2, 3 correspond to uniform meshes with $80 \times 32 = 2560, 160 \times 64 = 10,240, 320 \times 128 = 40,960$ linear finite elements.