Engineering Fracture Mechanics 115 (2014) 22–42

Contents lists available at ScienceDirect

Engineering Fracture Mechanics

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

Mixed-mode stress intensity factor evaluation by interaction integral method for quadratic tetrahedral finite element with correction terms

Ryutaro Daimon, Hiroshi Okada *

Department of Mechanical Engineering, Faculty of Science and Technology, Tokyo University of Science, 2641 Noda, Chiba 278-8510, Japan

ARTICLE INFO

Article history:
Received 9 May 2013
Received in revised form 31 August 2013
Accepted 6 November 2013
Available online 20 November 2013

Keywords:
Interaction integral method
Domain integral method
Stress intensity factors
Finite element method
Fracture mechanics

ABSTRACT

In this paper, a simple and accurate formulation of the interaction integral method for the quadratic tetrahedral finite element is presented. It was found in the course of present investigation that the auxiliary solutions set by the asymptotic solutions of the crack did not satisfy the equilibrium in terms of the finite element model consisting of the quadratic tetrahedral element. Thus, the results of the interaction integral computations contained a large magnitude of numerical error. To overcome this problem, the authors propose to add correction terms to the asymptotic solutions and to form new auxiliary solutions. The correction terms are determined so that the auxiliary solutions satisfy the equilibrium of the finite element model by performing finite element computations. Some numerical demonstrations are presented and they show that proposed methodology can give more accurate stress intensity factor solutions than the case without the correction terms.

1. Introduction

Stress intensity factor evaluation is the key process in a structural integrity analysis for a damaged structure with cracks due to fatigue or stress corrosion cracking (SCC) (see, for example, Atluri et al. [1] and Nakamura et al. [2]). Engineering structures are generally very complex in their configurations and the cracks often initiate at the locations of stress concentration as shown in the recent publication of Qian et al. [3] as an example. The stress analyses for such structures are generally carried out by the three-dimensional finite element method (FEM). When performing the FEM analysis in present computer hardware and software environment, we often use a three-dimensional solid modeler to define the model geometry. Then, we generate the FEM model using automatic meshing software and perform the three-dimensional FEM computation. However, the automatic model generation software is not able to generate an analysis model with cracks, in general. Therefore, when we perform the fracture analysis, the FEM model generation relies on our manual operations and takes a lot of man hours.

In last two decades, a series of works by many researchers have been presented to reduce the manual labor in the model generation processes. The meshless methods which are represented by the element-free Galerkin method (EFGM) (see, for example, [4–6]) totally eliminated the needs for meshing. Moving least square Petrov–Galerkin method (MLPG) was also proposed by Atluri and Zhu [7,8]. The EFGM and MLPG can perform analyses based only on nodal points. The extended finite element method (X-FEM) was proposed and applied to crack problems by Belytschko and Black [9] and Sukumar et al. [10].

* Corresponding author. Tel.: +81 4 7122 1808.
E-mail address: hokada@rs.noda.tus.ac.jp (H. Okada).

0013-7944/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.engfracmech.2013.11.009

Nomenclature

J^{3D}	three-dimensional J-integral
δA	area of virtual crack extension
V	region of domain integral for the domain integral method
W	strain energy density
δ_{ij}	Kronecker's delta
σ_{ij}	Cartesian components of the stresses
u_i	Cartesian components of the displacements
x_i	Cartesian coordinates
q_i	vector of virtual crack extension at the crack front/vector function for the domain integral method
$q(=\sqrt{q_i q_i})$	the absolute value of q_i
(r, s)	local coordinates to determine the value of q. r is the distance from the crack front. s is the coordinate in the tangent direction of the crack front.
q_{MAX}	maximum value of q
r_V	parameter to define the radius of region of domain integral
h_o, h_1	parameters to define the width of region of domain integral
K_I, K_{II}, K_{III}	mode I, II, II stress intensity factors
E, G and ν	Young's modulus, shear modulus and Poisson's ratio
$u_i'^{(2)}$	the displacements of auxiliary solutions that are defined as the two-dimensional asymptotic solutions
(r, θ)	local polar coordinates which are defined at the crack front
Δ_{Crack}	representative size (widths in tangential and normal directions of the crack front) of elements at the crack front
$\hat{\hat{u}}_i^{(2)}$, $\hat{\hat{\sigma}}_{ij}^{(2)}$	the correction terms of the auxiliary solutions for the displacements and stresses
$\hat{u}_i^{(2)}$, $\hat{\sigma}_{ij}^{(2)}$	the auxiliary solutions for the displacements and stresses with the correction terms
w_i	the weight functions to perform the FEM analysis to determine the correction terms $\hat{\hat{u}}_i^{(2)}$

In the X-FEM analysis, only the finite element mesh for uncracked structure needs to be provided by the analyst. The crack is inserted by enriching the interpolation functions by adding functions to represent jumps of the displacements across the crack faces and the $1/\sqrt{r}$ singular behavior of the stresses at the vicinity of the crack front. Thus, in the X-FEM analysis, the crack does not need to explicitly be modeled by the finite element mesh. Recently, isogeometric analysis which integrates CAD (Computer Aided Design) and the finite element method attracted many researchers. The readers are referred to Cottrell et al. [11]. The isogeometric analysis was combined with the concept of the X-FEM by Ghorashi et al. [12]. In Ghorashi et al. [12], two-dimensional crack propagation analyses were successfully demonstrated. The s-version finite element method (S-FEM) that superposes a finite element model to represent some local feature such as crack on that representing the global structure was proposed by Fish [13] and was later applied to two-dimensional crack problems by Okada et al. [14]. S-FEM is a useful tool to perform complex three-dimensional crack propagation analyses as presented by Kamaya et al. [15]. These methodologies totally obviate or drastically reduce the meshing tasks in performing the crack analyses.

Crack propagation analyses by using the conventional finite element method are also found in literature. SchÖllmann et al. [16] developed a software system called ADPCRACK3D which can perform crack propagation analysis in a complex three-dimensional structure. The global structure may be modeled by the tetrahedral finite elements but the crack and its surroundings are discretized by the hexahedral elements using a submodeling technique. FRANC3D/NG [17] was developed by a group of researchers at Cornell University. FRANC3D/NG can deal with cracks in general three-dimensional structures by using a combined modeling methodology in which the structure as whole and the vicinity of the crack front are modeled by the tetrahedral and by the hexahedral elements, respectively. Bremberg and Dhondt [18] also reported the combined modeling methodology. The combined element methodologies are developed so that proven numerical techniques to evaluate the crack parameters using the hexahedral finite element can be adopted. Lucht [19] presented a combined finite element/boundary element methodology. The boundary element sub-model is placed at the vicinity of the crack but the structure as whole is modeled by the tetrahedral finite elements.

On the other hand, the meshing procedures for the tetrahedral elements may be much simpler than those for the combined approaches. As described in a book chapter of Zienkiewicz et al. [20], the automatic meshing methodologies based on the advancing front method and the Delaunay triangulation are proven techniques and the processes of finite element model generation can fully be automated when the tetrahedral elements are adopted. Such methodologies for general 3D-structures are already in practical and commercial use (see, for example, Technostar Co., Ltd. [21]). For fracture mechanics analyses, Okada et al. [22,23] and Kaneko et al. [24] presented relatively simple automatic mesh generation schemes for arbitrary shaped cracks in complex three-dimensional structures. To the authors' best knowledge, the reasons why the hexahedral elements are commonly adopted in the fracture mechanics analyses are that (i) the algorithms to