International Journal of Solids and Structures 62 (2015) 144–157

Contents lists available at ScienceDirect

International Journal of Solids and Structures

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

A numerical procedure for interaction integrals developed for curved cracks of general shape in 3-D

Daniel Bremberg *, Jonas Faleskog

Department of Solid Mechanics, Royal Institute of Technology (KTH), 100 44 Stockholm, Sweden

ARTICLE INFO

Article history:
Received 5 August 2013
Received in revised form 15 December 2014
Available online 25 February 2015

Keywords:
3-D
Mixed-mode
Curved cracks
Interaction energy integral
Stress intensity factors

ABSTRACT

This study presents a numerical procedure for the evaluation of interaction energy integrals used to extract mixed-mode stress intensity factors. The interaction energy integral is expressed as a domain integral, and the proposed numerical procedure delivers accurate results for three-dimensional cracks with curved crack fronts and curved crack surfaces for a rather general set of integration domains. It is clearly shown that, when the curvature of the crack surface becomes sufficiently large, special care must be taken in the evaluation of both the volume and the area integrals involved. To improve the accuracy in the evaluation of the former, a composite rule for the Gaussian quadrature scheme is employed. Four benchmark geometries with available analytical solutions are considered. Firstly, mesh design parameters for planar cracks with straight and curved crack fronts are established. Secondly, non-planar cracks with straight and curved crack fronts are employed to examine the accuracy of the numerical procedure.

1. Introduction

Fatigue crack growth computations in three dimensions (3-D) have become a necessity in many applications in order to meet the demands put on advanced engineering structures with high safety requirements, e.g. nuclear and aerospace industries. Applications and algorithms for computations in two dimensions (2-D) have over time evolved into stable and reliable frameworks. However, the transition from 2-D to 3-D analyses has proven to be difficult and still concerns the community for a wide range of applications. 3-D fracture computations involve both numerical and theoretical challenges. The theoretical challenges focus on the development of fracture criteria and evolution laws that are as valid in a 3-D context as they are in a 2-D context. This study focuses on some of the numerical issues.

Numerical algorithms for fracture computations in 3-D are inherently difficult due to the problem of setting up robust automatic meshing procedures. The numerical problems arise in the preprocessing step during mesh design and in the postprocessing step during calculations of the stress intensity factors. Modeling difficulties are typically common for complex geometries, but they appear also for simple structures with non-planar cracks. It is generally advised to generate a focused and well-structured mesh at the crack front location. The task is fairly straightforward in 2-D but it is non-trivial in 3-D where the added depth has proven particularly demanding. Some frameworks ignore the general advice and experiment with an unstructured mesh, usually a tetrahedral mesh. This simplifies the preprocessing work but it makes the postprocessing more difficult. The postprocessing scheme depends on the ability to identify elements located in connection to the crack front location, relative location, and their connectivity. In cases of well-structured and focused meshes, this becomes easy since it only requires a simple book-keeping step. However, if no information about the mesh arrangement is provided to the postprocessor, calculations of the stress intensity factors may become highly demanding. A well-documented element topology makes it possible to adopt virtually any postprocessing method available but not necessarily otherwise.

There are a number of 3-D fatigue crack growth software applications that over the years have been developed based on the finite element method (FEM) framework (see for instance Dhondt, 1998; Schöllmann et al., 2003; Carter et al., 2000; Zencrack, 2013 and FEACrack, 2013). The software usually consists of a pre- and postprocessor that has an interface to one or more FE solvers (see for instance Abaqus, 2013; ANSYS, 2013; CalculiX, 2012 and Warp3D, 2012).

Bremberg and Dhondt (2008, 2009) developed a methodology for introducing cracks of arbitrary shape into existing FEM models of engineering structures. Their methodology builds on identifying a subregion in which a crack is to be inserted. The crack front is independently modeled by a focused tubular mesh along the crack

* Corresponding author.
E-mail address: bremberg@kth.se (D. Bremberg).

http://dx.doi.org/10.1016/j.ijsolstr.2015.02.022
0020-7683/

front consisting of 20-node tri-quadratic 3-D hexahedral elements. The domain between the tubular mesh and the boundaries of the subregion is then closed by meshing the domain with 10-node quadratic 3-D tetrahedral elements. This methodology is illustrated in Fig. 1, which shows a turbo-charger compressor (CalculiX, 2012) in which a crack has been inserted. The tubular region with structured hexahedral mesh around the crack front, highlighted in blue, is most visible in the close-up picture in Fig. 1(c). The remaining part of the subregion, consisting of tetrahedral elements, is shown in yellow. The tubular mesh around the crack front should preferably be optimized to facilitate accurate estimations of parameters that characterize the crack-tip state, e.g. the stress intensity parameters in a linear elastic solid. Several methods exist to extract such parameters from the solutions of a finite element analysis. An accurate and efficient method for this purpose is the interaction energy integral method, which was employed in this study.

Interaction energy integrals, originating from a contour integral (2-D) by Stern et al., 1976, were developed into a 3-D formulation by Shih and Asaro (1988) within the framework of domain integration (Li et al., 1985; Shih et al., 1986). The same integral concept can also be used to determine non-singular crack-tip parameters such as the *T*-stress (Nakamura and Parks, 1992). Gosz et al. (1998) and Gosz and Moran (2002) generalized the interaction energy integral method to account for non-planar crack surfaces with curved crack fronts. The general formulation involves additional surface integrals that can be difficult to evaluate with satisfactory accuracy. The importance of these surface integrals vanishes as the integration domains become sufficiently small, and this feature is often utilized in practice (Walters et al., 2005). However, this only works well when the encompassed portion of a crack surface is approximately planar. Numerical fatigue crack growth analysis requires a large number of successive crack fronts to be evaluated incrementally. Due to gradients in the stress and strain fields, a fatigue crack may advance with deflection angles and increment lengths varying along each evaluated crack front. Hence, relatively small integration domains may encompass highly irregular and non-planar crack surfaces for cracks grown by fatigue within a discretized modeling framework.

The objective of the current work is to address numerical issues that arise when evaluating domain integrals for stress intensity factor (SIF) calculations along an arbitrary 3-D crack front, including severe crack surface curvature. This is accomplished by first scrutinizing the design of a tubular mesh along an arbitrary 3-D crack front in order to give accurate SIF solutions with a reasonable computational effort. For this purpose, a set of benchmark examples was used in this study. A through crack in a finite plane structure (2-D) is used for cross-sectional mesh design. Embedded planar circular and elliptical cracks are used to investigate the effect of curved crack fronts with constant and variable crack front curvature. The analysis is then extended to non-planar crack surfaces by employing an arc crack and a lens crack.

2. Numerical framework for linear elastic crack analyses

The methodology developed by Bremberg and Dhondt (2008, 2009) will now be briefly summarized. Instead of adapting the crack front domain to fit a given configuration, the method presented here is based on a slightly different approach. There are a couple of remarks about the underlying ideas that first have to be made before describing the essentials of the algorithm. Crack growth is governed and described by the loading state at the crack front. Consequently, a suitable mesh in the specified neighborhood must be adequately designed in order to enable correct and accurate computations of stress intensity factors and the resulting crack growth. The common belief is that this requires a structured and focused mesh at the crack tip location, which has long been the standard approach for 2-D computations within the framework of the FEM. However, the introduction of a third dimension has proven non-trivial. This is particularly valid for the automatic insertion of a crack into an initially uncracked mesh.

The framework is based on three separate modules that together constitute a crack growth increment computation: a preprocessor, an FE solver and a postprocessor. The FE solver adopted here is that presented by CalculiX (2012). Crack-growth analyses are by nature incremental processes that are performed for a selected interval of load sequences. The computation is usually terminated as the analysis reaches the end of the specified interval or at a point where critical or zero crack growth is reached. A typical crack growth analysis most often involves a large number of increments, which is why automation of the procedure is necessary.

The proposed preprocessor methodology uses the crack front as the starting point for the crack insertion. The crack surface and the free boundary of the portion of the initially uncracked structure that will hold the crack are identified and represented by triangular meshes. A system of local crack tip coordinate systems is automatically set up along the crack front by which a tubular interface can be defined, with the crack front acting as the trajectory line. After the tubular interface has been adapted to the free boundary of the uncracked structure and the crack surface, it is automatically filled with a structured and focused hexahedral mesh with collapsed quarter point elements in immediate connection to the trajectory line. A geometrical description of the volume of the selected portion of the initially uncracked structure is

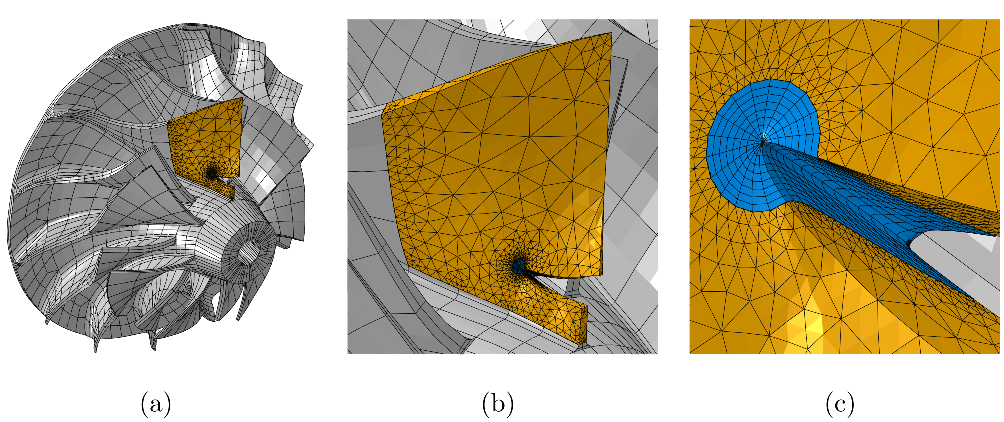

Fig. 1. A selected blade from (a) a turbo-charger compressor is used to illustrate (b) the transition mesh in yellow and (c) the tubular mesh in blue. The displacements have been magnified for illustration purposes. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)