Available online at www.sciencedirect.com
SCIENCE DIRECT®

Engineering Fracture Mechanics 73 (2006) 490–504

Engineering Fracture Mechanics

www.elsevier.com/locate/engfracmech

A methodology for automatic crack propagation modelling in planar and shell FE models

D. Colombo [*], M. Giglio

Dipartimento di Meccanica, Politecnico di Milano, 20156 Milano, Italy

Received 9 February 2005; received in revised form 8 August 2005; accepted 16 August 2005
Available online 21 October 2005

Abstract

This paper describes a fast and reliable algorithm for automatic simulations of crack propagation in bi-dimensional and non-planar shell FE models. Thanks to its simplicity, the algorithm can be coded with little effort and in a relatively short time. Moreover, it can be interfaced with the existing FE analysis environment. An automatic iterative process makes it possible to achieve the highest degree of accuracy in the results for the FE model used. Good accuracy can also be achieved for quite coarse meshes.

Keywords: Crack propagation; Fatigue; Fracture mechanics; Finite element

1. Introduction

Nowadays, crack propagation problems are usually solved by means of the finite element method. The propagation process is divided into several short steps. In each one, the stress intensity factors (SIF) are evaluated and the direction of propagation is calculated using the propagation criterion that is best suited to the particular problem. Finally, the FE model is modified in order to accommodate a short, straight propagation of the crack in the direction evaluated.

The whole process is repeated many times and is both time-consuming and a source of errors if done manually. This is particularly true for the modification of the mesh. Software capable of handling the solution process in an automated way is welcome.

The software should address three different aspects of the problem: evaluation of the stress intensity factors, evaluation of the direction of propagation and modification of the mesh to accommodate crack advancement. The first and second aspects can be solved with no particular difficulties and many commercial FE codes have built-in capabilities to evaluate the SIF. The last is much more complex and is rarely solved by currently available FE codes, e.g. [1,2].

---

[*] Corresponding author. Tel.: +39 02 23998213; fax: +39 02 23998202.
E-mail address: daniele.colombo@polimi.it (D. Colombo).

0013-7944/$ - see front matter © 2005 Elsevier Ltd. All rights reserved.
doi:10.1016/j.engfracmech.2005.08.007

Different approaches have been used to solve this problem. The simplest is the so-called 'debonding technique' [3,4]: the propagation is simulated by disconnecting the nodes in common between two elements. In this way, the crack can propagate only along the edges of the elements in the mesh. This implies that the propagation path be known at the beginning of the simulation in order to obtain accurate results. This is not usually the case.

A more complex approach is based on remeshing [3,5–8]: the whole model or part of it is remeshed at each step of the propagation in order to accommodate the new geometric configuration of the structure. This technique gives accurate results but has some disadvantages. The main one is that available remeshing algorithms are quite complex and enormous effort and advanced skills are required of the programmer to write a functional code. Another disadvantage is that the time and the computational resources required for remeshing for geometrically complex structures are considerable. This disadvantage can, however, be minimized by limiting the remeshing to a small area surrounding the crack tip. In any case, there is still the problem of connecting the newly generated mesh to the existing one. The conformity requirement between the two meshes adds a further difficulty to the problem. The remeshing approach can also be problematic in cases where data or part of the solution need to be transferred between the models of two consecutive steps of the propagation, for example in the common situation of the presence of residual stresses in the structure. This is not a complex task but the accuracy of the transferred data can degrade with the propagation due to the fact that the meshes between two consecutive steps are different and an approximation in the transfer process is required.

This paper presents a simple but effective solution for any bi-dimensional and shell model. The algorithm has been developed with simplicity and speed in mind. It can be coded quite quickly by anyone with a basic programming background and using any programming language. It can manage multiple cracks in the structure at the same time. It can be used in conjunction with the existing finite element analysis environment provided that the FE solver has submodelling capabilities [9,10]. Thus, for fracture mechanic simulations, too, the analyst can thus adopt the FE solver he or she usually uses and with which he or she feels comfortable. Moreover, this solution allows the use of existing FE models. In fact, in many cases, fracture mechanic analysis is only a part of the whole design process and different kinds of numerical structural analyses need to be carried out on the same part of structure. In such a scenario, the use of a single FE model for all the analyses is very useful. Of course, this model should be developed in such a way as to be suitable for all the analyses that need to be performed.

The algorithm is very fast for complex and large FE models, too. The accuracy of the results is obviously dependent on the initial mesh of the model. However, even with quite coarse meshes, errors obtained can be small. This is undoubtedly positive because it eliminates the need for an initial FE model which is highly refined in the area of crack propagation. The accuracy problem is solved partially by the algorithm, as set out later in the paper. In such a manner, the dimensions of the model can be kept relatively small without any loss of accuracy in the solution.

2. The propagation algorithm

The idea on which the algorithm is based is very simple. At each step of the propagation, an FE model is already defined and the algorithm can work on it: at the first step, the model is provided by the analyst as input for the simulation; in the following steps, the model is provided by the output of the algorithm itself in the previous step. In each step, only a few elements of the existing model are affected by the propagation. They are the ones that lie on the straight path followed by the crack in that step. As a consequence, the model can be updated for the next step simply by modifying these elements. There is no need to remesh the entire model or a small area surrounding the crack and the upgrade of the model can be obtained by splitting the elements involved in the propagation. This can be done by replacing the element in a well-defined manner, as explained later.

An example illustrates how this basic idea works. Suppose that the situation at a generic step of a simulation is the one depicted in Fig. 1a, in which only the area of the model near the crack tip is shown. The faces of the crack have been drawn exaggeratedly apart for the sake of clarity. The direction and magnitude of propagation is represented by the vector in the figure. The elements of the mesh affected by the propagation are shaded. Only these elements must be modified in order to accommodate the propagation. They are removed