Available online at www.sciencedirect.com
ScienceDirect
ELSEVIER
Computational Materials Science 39 (2007) 1–7
COMPUTATIONAL MATERIALS SCIENCE
www.elsevier.com/locate/commatsci

A comparison of micro, meso and macroscale FEM analysis of ductile fracture in a CT specimen (mode I)

Sanjeev Saxena, N. Ramakrishnan *

Regional Research Laboratory, Hoshangabad Road, Bhopal 462026, India

Received 14 October 2005; received in revised form 16 December 2005; accepted 18 January 2006

Abstract

Several methods are available to understand the process of crack initiation and propagation in ductile materials. In an attempt to achieve an overall understanding, some of these techniques were studied using a large deformation based finite element method (FEM). In the current investigation, typical crack tip blunting prior to ductile fracture behavior of a standard (CT) specimen under mode I loading condition was simulated using FEM. An attempt was made to understand the ductile fracture by numerically determining the ductile fracture toughness at three length scales: macroscopic scale (load–displacement method), mesoscopic scale (path-integral method) and microscopic scale (stretch zone width method). In addition, the characteristic distance (l_c), commonly defined as the distance between the crack tip and the void responsible for eventual coalescence with the crack tip, was also studied. Although approximate, l_c assumes a special significance since it links the fracture toughness to the microscopic mechanism considered responsible for ductile fracture.

Keywords: Ductile fracture; Finite element method; Fracture toughness; Characteristic-distance

1. Introduction

In ductile materials, fracture is understood to occur by void nucleation at a distance from the crack tip and subsequent coalescence with the crack tip [1]. When a material with a crack is loaded in tension, the deformation energy builds up around the crack tip and it is understood that at a certain critical condition voids are formed ahead of the crack tip. The crack extension occurs by coalescence of voids with the crack tip. A fracture criterion that could accurately predict failure would be a useful engineering tool both for the evaluation of structural integrity and the selection of materials. Complex structures may experience stress in some regions that exceed the elastic limit necessitating a fracture criterion that would also include elastic–plastic behavior. The path independent J integral proposed by Rice [2] is a well-established parameter that describes the crack tip elastic–plastic field.

Several attempts had been made to understand the process of crack initiation and propagation in ductile materials. Landes and Begley [3–5] had presented an experimental means to predict the fracture toughness, which is popularly referred to as load–displacement method. Paranjpe and Banerjee [6], Mills [7], Amouzouvi and Bassim [8], Yin et al. [9], Doig et al. [10] and Bassim et al. [11] demonstrated experimentally the procedure to evaluate the fracture toughness using stretch zone width (SZW) measurement. Kobayashi et al. [12] compared the standard procedure of determining the fracture toughness to the one based on SZW. Bassim [13] and Smith et al. [14] discussed SZW for fracture toughness measurement. Tai [15] employed FEM route to study the damage ahead of the crack tip. Using crack tip opening displacement (CTOD) and SZW parameter measurement, Ebrahimi and Seo [16] discussed the process of crack initiation. Wang and Hwang [17] used crack tip opening angle and J-integral to understand the experimental test data of CT specimen.

* Corresponding author. Tel.: +91 755 2587105; fax: +91 755 2587042.
E-mail addresses: ramkrish@rrlbpl.org, ramkrish@sify.com (N. Ramakrishnan).

0927-0256/$ - see front matter © 2006 Elsevier B.V. All rights reserved.
doi:10.1016/j.commatsci.2006.01.022

Knott [18], Garrison Jr. [19], Ritchie and Thompson [20] related the characteristic distance to microstructural parameters such as grain size, inter-particle distance etc. Tsann Lin et al. [21] analyzed the effect of grain size as well as particle spacing on fracture toughness. Srinivas et al. [22] established l_c experimentally in Armco iron, relatively a 'clean material', in order to assess its dependency on grain size or particle spacing and showed that the voids could occur well inside the grains as well demonstrating that a grain boundary or particle boundary is not necessarily a pre requisite for the void formation.

In the present work, deformation behavior of CT specimens of Armco iron under mode I loading condition is simulated using finite element method. In an attempt to compare different methods of characterizing ductile fracture, the fracture toughness (J) is determined numerically at three length scales: macroscopic scale (load–displacement method), mesoscopic scale (path-integral method) and microscopic scale (stretch zone width method). In addition, an attempt has been made to establish the characteristic distance (l_c) correlation with fracture toughness. This method assumes a special significance since it links the fracture toughness to the microscopic mechanism of ductile fracture. The experimentally [22] obtained properties of Armco iron for different grain sizes are used as the effective properties of the homogeneous continuum in the present study. The study essentially pertains to ductile crack initiation and is not concerned with crack growth related aspects. Further, FEM analysis is used more as a numerical experiment and therefore the specimen geometry, the material properties and the boundary conditions were so chosen as to conform to the experimental conditions [22]. This would allow validation of the numerical study as well. While J can be assessed using numerical methods, to obtain the critical J, the critical displacement of 1.5 mm as observed in the experiments [22] was imposed here.

2. Numerical model

2.1. Geometry

The investigation was limited to compact tension (CT) specimen subjected to mode I type of loading and the FEM mesh model of standard CT is shown in Fig. 1 [23]. The symmetry in this case permits consideration of only one half of the specimen geometry for computational economy. For modeling details of the fracture specimen, refer to Ramakrishnan et al. [24]. Various investigators successfully employed dense mesh with conventional elements and advocated against the use of singular or special crack tip elements unless required specifically [25]. Following the above guidelines, the mesh was constructed with a set of bi-linear four-noded quadrilateral elements. Since the experimentally measured characteristic distance (l_c) was about 100 μm [22], the size of the crack tip elements was chosen to be about 5 μm and increased upto 10,000 μm (1 cm) radially in a geometric progression as suggested by Tsamaphyros and

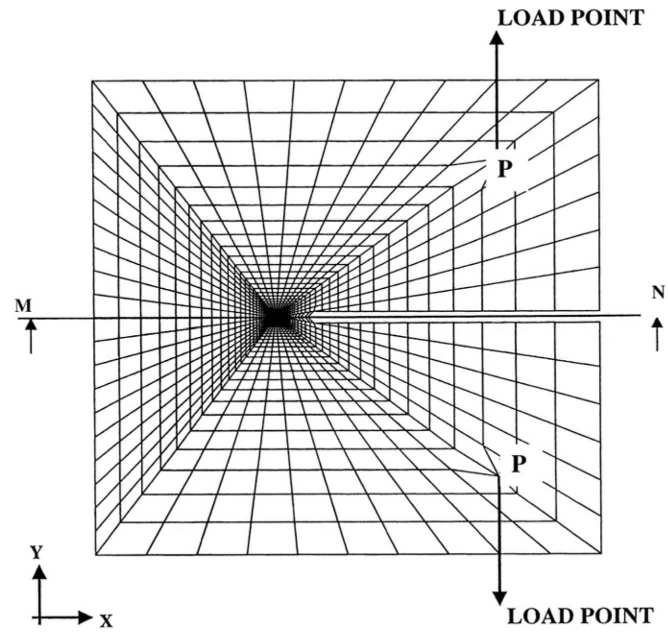

Fig. 1. Finite element mesh for compact tension specimen.

Glannakopoulos [26]. This type of mesh configuration allowed the study of deformation at the microscopic scale in the vicinity of the crack tip as well as determination of the global load–displacement variation for the entire specimen. Although, at a level of 5 μm mesh size near the crack tip, conditions are no longer isotropic due to different crystal orientations but the average behavior is assumed to be polycrystalline isotropic with random grain orientations. Since we are not addressing any structure sensitive issues such as grain boundary or particle boundary, the assumption can be considered appropriate. A magnified view of the mesh morphology around the crack tip is shown in Fig. 2a and that of the corresponding deformed mesh in Fig. 2b. Varying the total number of elements from 500 to 2000 and examining the convergence of the numerical results, an optimum of 1300 elements was arrived at. The structured mesh was used in all the numerical simulations for the post processing-convenience it offers.

2.2. Material property

The material undergoes large strain and rotation at the crack tip necessitating a constitutive framework based on finite deformation for the numerical simulation. Accordingly the present investigation used a finite deformation algorithm [27] based on total-elastic-incremental-plastic strain that was originally developed for simulating metal forming processes where the local deformation is similar to that of crack tip blunting. The general aspects of the large deformation algorithm have been validated with a number of analytical and experimental data [27], and the validations specific to the present investigation was carried out with reference to the published numerical [28] and the experimental results [22]. The details are given in [24].