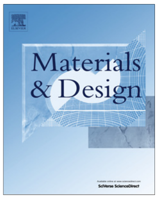
Materials and Design 55 (2014) 191–196

Contents lists available at ScienceDirect

Materials and Design

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



Application of extended finite element method in damage progress simulation of fiber reinforced composites

H.W. Wang [a,*], H.W. Zhou [b], H.W. Ji [c], X.C. Zhang [c]

[a] *Tianjin Key Laboratory of Refrigeration Technology, Tianjin University of Commerce, Tianjin 300134, China*
[b] *State Key Laboratory of Coal Resources and Safe Mining, China University of Mining and Technology, Beijing 100083, China*
[c] *School of Mechanical Engineering, Tianjin University of Commerce, Tianjin 300134, China*

ARTICLE INFO

Article history:
Received 9 July 2013
Accepted 29 September 2013
Available online 10 October 2013

Keywords:
Extended finite element method
Damage progress simulation
Fiber reinforced composites
Convergence issues

ABSTRACT

Damage initiation and propagation in unidirectional glass fiber reinforced epoxy matrix composites under tension load were simulated in this study. Cell models with both single fiber and multi-fiber were modeled by extended finite element method (XFEM) in ABAQUS environment. The damage progresses in the cells were investigated and then the nominal stress–strain curves as well as stress distributions in the fiber and matrix were obtained. Results indicate that the XFEM is an effective modeling technique to study the initiation and propagation of a crack along an arbitrary, mesh-independent, solution-dependent path. In addition, convergence difficulties are familiar issues while carrying out damage and fracture analysis when employing numerical simulation. Several methods, which are helpful to resolve these convergence problems, were presented. Baseline simulations about damage initiation and propagation in composites indicate that a larger value of viscous regularization causes the peak of the reaction force to be higher. A smaller viscosity parameter is better than bigger one. Other advanced techniques, include using automatic stabilization and customized general solution controls, are also adoptable to improve convergence in ABAQUS/Standard analysis.

Crown Copyright © 2013 Published by Elsevier Ltd. All rights reserved.

1. Introduction

Glass fiber-reinforced polymer (GFRP) composites are widely used in the low-weight constructions, due to their high strength of glass fibers as well as the availability of efficient and low cost production technologies of the materials. Unidirectional composites with epoxy matrixes, which have better mechanical properties than the polyesters and vinyl resins, are often used in the aerospace and wind energy applications. In response to these requirements, research on fiber reinforced composites has attracted much attention in mechanics and materials science fields. This has led to many publications with new information of fiber reinforced composites in wide fields. For example, Mishnaevsky did a systematic numerical analysis of the effect of microstructures of composites on their strength and damage resistance [1]. Lei et al. investigated the local buckling couple with the whole bending of the composite panel by digital fringe projection profilometry [2]. Cen et al. carried out research of effect of geometry on interfacial micromechanical behavior of fiber/matrix microdroplets by means of the combination of micro-bond test and Micro-Raman spectroscopy [3]. Kang et al. developed an inverse/genetic method for interfacial parameter identification for composites [4]. Wang et al. presented a three-parameter interfacial model based on Needleman's cohesive theory to characterize the viscoelastic mechanical properties of composites [5]. Zhou et al. carried out experimental investigations on damage growth in GFRP under three-point bending tests [6] and analysis of compressive damage mechanism of GFRP under off-axis loading [7]. Li and Qiu et al. investigated the deformation mechanisms of carbon nanotube fibers under tensile loading by in situ Raman spectroscopy analysis [8,9].

In practice, components made from fiber reinforced composites may eventually fail because of pre-existing defects such as interface debonding, micro-cracks and voids. The reliability of the components made from fiber reinforced composites may be predicted and eventually increased on the basis of the analysis of the effect of the composite microstructure on the strength and damage evolution. Such an analysis can be carried out in the framework of computational experiments.

In order to evaluate the mechanical behaviors of composites materials, different approaches, including experimental investigation [10,11], numerical simulations [1] and theoretical modeling [12,13], were employed. To predict the strength and other properties of composites, a number of mathematical models of deformation, damage and failure of fiber reinforced composites have been developed. As reviewed by Mishnaevsky [1,14], these models are classified into several groups as: numerical continuum mechanical models [15,16], shear lag-based analytical models [17], fiber

---

[*] Corresponding author. Tel.: +86 22 2668 6251; fax: +86 22 2668 6268.
E-mail address: wanghw@tjcu.edu.cn (H.W. Wang).

0261-3069/$ - see front matter Crown Copyright © 2013 Published by Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.matdes.2013.09.071

bundle model and its generalizations [18], continuum damage mechanics based models [19] and fracture mechanics based models and crack bridging [20], stress transfer theoretical model [21], etc. More information on the models of damage and fracture of fiber reinforced composites can be found elsewhere [1].

The numerical continuum mechanical models, such as finite element model, allow the incorporation of many different features of the nonlinear material behaviors. They are computational technique for obtaining approximate solutions to the partial differential equations that arise in scientific and engineering applications [22,23]. The advantages of the finite element method over the finite difference method are that general boundary conditions, complex geometry, and variable material properties can be relatively easily handled. Finite element analysis is at present very widely used in the engineering analysis of heat transfer, mass diffusion, thermal management of electrical components (coupled thermal–electrical analyses), acoustics, soil mechanics (coupled pore fluid-stress analyses), and piezoelectric analysis. As a universal finite element method commercial software, ABAQUS has been widely used in scientific research and engineering applications [24,25]. However, convergence difficulties are familiar issues while carrying out damage and fracture analysis in ABAQUS/Standard. Such as: (1) in many cases, cohesive elements are modeled as undergoing progressive damage leading to failure. The modeling of progressive damage involves softening in the material response, which is material nonlinear; (2) if using other facture criterion, such as VCCT (virtual crack closure technique), convergence difficulties may also occur during unstable crack propagation [26].

In this work, numerical continuum mechanical models were used to simulate the damage progress in unidirectional glass fiber reinforced composites under tension load. Cell models with multi-fiber and with single fiber were modeled using XFEM. In addition, some discussion on resolving convergence issues, including using viscous regularization, using automatic stabilization and using nondefault solution controls, were given. Moreover, discussion on the effect of using viscous regularization on the results was also presented. The results of the simulation are expected to provide some design parameters for the optimal performance of these glass fiber reinforced composites.

2. Extended finite element method (XFEM)

In recent years, the extended finite element method (XFEM) has emerged as a powerful numerical procedure for the analysis of crack problems [26,27]. In comparison to the conventional finite element method, the XFEM provides significant benefits in the numerical modeling of crack propagation. In the conventional formulation of the FEM, the existence of a crack is modeled by requiring the crack to follow element edges. In contrast, the crack geometry in the XFEM need not be aligned with the element edges. XFEM can be used to study the initiation and propagation of a crack along an arbitrary, mesh-independent, solution-dependent path, which provides flexibility and versatility in modeling [26,27].

Phantom nodes, which are superposed on the original real nodes, are introduced to represent the discontinuity of the cracked elements, as illustrated in Fig. 1. When the element is intact, each phantom node is completely constrained to its corresponding real node. The cracked element splits into two parts when the element is cut through by a crack. Each phantom node and its corresponding real node are no longer tied together and can move apart [26].

For the purpose of fracture analysis, the enrichment functions typically consist of the near-tip asymptotic functions that capture the singularity around the crack tip and a discontinuous function that represents the jump in displacement across the crack surfaces.

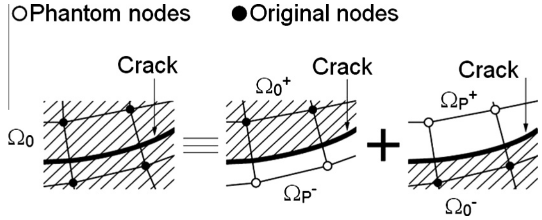


Fig. 1. The principle of the phantom node method.

The approximation for a displacement vector function with the partition of unity enrichment is

$$u = \sum_{i=1}^{N} N_i(x)\left[u_i + H(x)a_i + \sum_{\alpha=1}^{4} F_\alpha(x) b_i^\alpha\right] \tag{1}$$

where $N_i(x)$ are the usual nodal shape functions; the first term on the right-hand side of the above equation, u_i, is the usual nodal displacement vector associated with the continuous part of the finite element solution; the second term is the product of the nodal enriched degree of freedom vector, a_i, and the associated discontinuous jump function $H(x)$ across the crack surfaces; and the third term is the product of the nodal enriched degree of freedom vector, b_i^a, and the associated elastic asymptotic crack-tip functions, $F_\alpha(x)$. The first term on the right-hand side is applicable to all the nodes in the model; the second term is valid for nodes whose shape function support is cut by the crack interior; and the third term is used only for nodes whose shape function support is cut by the crack tip [26].

Most of the fiber reinforced composites materials exhibit elastic–brittle behavior. There is less significant plastic deformation before damage initiation. A typical stress–strain relation is mostly linear-elastic in a first part and then tracked by degradation until the material fully losses its stiffness, which is called traction-separation response. In this study, bilinear model was employed to describe the pre- and post-damage process of a traction-separation, as shown in Fig. 2. Damage initiation refers to the beginnings of the degradation of the response of a material point. The process of degradations begins when the stresses and/or strains satisfy certain damage criterion. Here maximum nominal stress criterion was employed. Once the damage initiation criterion is reached, the damage evolution describes the rate of degradation of the material. A scalar damage variable D is introduced to represent the overall damage process. D monotonically evolves from 0 (no damaged has occurred) to 1 (overall damaged) upon further loading after damage initiation.

3. Damage initiation and propagation in composites

In order to explore the damage initiation and propagation in GFRP, models with different fiber number were developed. The

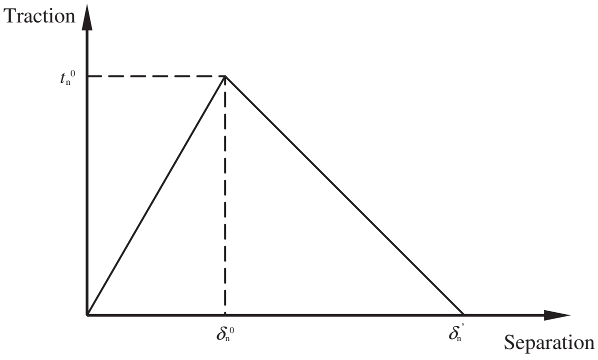


Fig. 2. A typical bilinear traction-separation model.

Download English Version:

https://daneshyari.com/en/article/829525

Download Persian Version:

https://daneshyari.com/article/829525

Daneshyari.com