Computational Materials Science 46 (2009) 810–820

Contents lists available at ScienceDirect

Computational Materials Science

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

Single fibre and multifibre unit cell analysis of strength and cracking of unidirectional composites

H.W. Wang [a,b], H.W. Zhou [b], L. Mishnaevsky Jr. [c,*], P. Brøndsted [c], L.N. Wang [b]

[a] School of Mechanical Engineering, Tianjin University of Commerce, Tianjin 300134, China
[b] Institute of Rock Mechanics and Fractals, China University of Mining and Technology (Beijing), Beijing 100083, China
[c] Risø DTU National Laboratory for Sustainable Energy, Technical University of Denmark, AFM-228, P.O. Box 49, Frederiksborgvej 399, DK-4000 Roskilde, Denmark

ARTICLE INFO

Article history:
Received 2 March 2009
Received in revised form 2 April 2009
Accepted 15 April 2009
Available online 15 May 2009

PACS:
81.05.Qk
02.70.Dh
62.20.Mk

Keywords:
Micromechanical modelling
Fibre reinforced composites
Damage evolution
Cohesive elements
Finite element method

ABSTRACT

Numerical simulations of damage evolution in composites reinforced with single and multifibre are presented. Several types of unit cell models are considered: single fibre unit cell, multiple fibre unit cell with one and several damageable sections per fibres, unit cells with homogeneous and inhomogeneous interfaces, etc. Two numerical damage models, cohesive elements, and damageable layers are employed for the simulation of the damage evolution in single fibre and multifibre unit cells. The two modelling approaches were compared and lead to the very close results. Competition among the different damageable parts in composites (matrix cracks, fibre/matrix interface damage and fibre fracture) was observed in the simulations. The strength of interface begins to influence the deformation behaviour of the cell only after the fibre is broken. In this case, the higher interface layer strength leads to the higher stiffness of the damaged material. The damage in the composites begins by fibre breakage, which causes the interface damage, followed by matrix cracking.

1. Introduction

Fibre reinforced composites find wide applications in aerospace and marine industry, wind power, automobile, civil engineering, national defence, etc., due to their high strength, low weight and high toughness. Research on fibre reinforced composites has attracted much attention in mechanics and materials science fields, and led to many publications with new information of fibre reinforced composites in wide fields.

In practice, components made from fibre reinforced composites may eventually fail because of pre-existing defects such as microcracks, voids and interface debonding. The reliability of the components from fibre 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 [1–3].

In order to model the damage and failure of fibre reinforced composites under mechanical load, several modelling approaches are employed [1]: the shear lag and other analytical models, used to analyze the load transfer and multiple cracking in composites [4–7], fibre bundle model [8,9], fracture mechanics-based models [10–12], and numerical continuum mechanical models [1–3,13–18]. The numerical continuum mechanical models, usually finite element models, allow the incorporation of many different features of the nonlinear material behaviours and the analysis of the interaction of available and evolving microstructural elements [1–3,13].

In this work, we used numerical continuum mechanical models to simulate the damage evolution, interaction between different damage mechanisms, and the effect of the phase and interface properties in unidirectional glass fibre reinforced composites under tension load. The results from cohesive zone modelling, and the modelling based on the damageable layer with element weakening technique were compared. The results of the simulation are expected to provide some design parameters for the optimal performance of these glass fibre reinforced composites.

2. Generation of unit cell models of composites with damageable elements

2.1. 3D unit cells with single and multiple fibres

A number of unit cell models of composites were generated with the use of the program code "Meso3DFibre" [1–3]. The

* Corresponding author. Tel.: +45 4677 5729; fax: +45 4677 5758.
E-mail address: lemi@risoe.dtu.dk (L. Mishnaevsky).

0927-0256/$ - see front matter © 2009 Elsevier B.V. All rights reserved.
doi:10.1016/j.commatsci.2009.04.024

program generates a command file for the commercial FE pro- and post-processing software MSC/PATRAN©, which produces a 3D unit cell model of composite with pre-defined parameters. The 3D finite element meshes are generated by sweeping the corresponding 2D meshes on the surface of the microstructure models. The program "Meso3DFibre" is described in more details elsewhere [1–3].

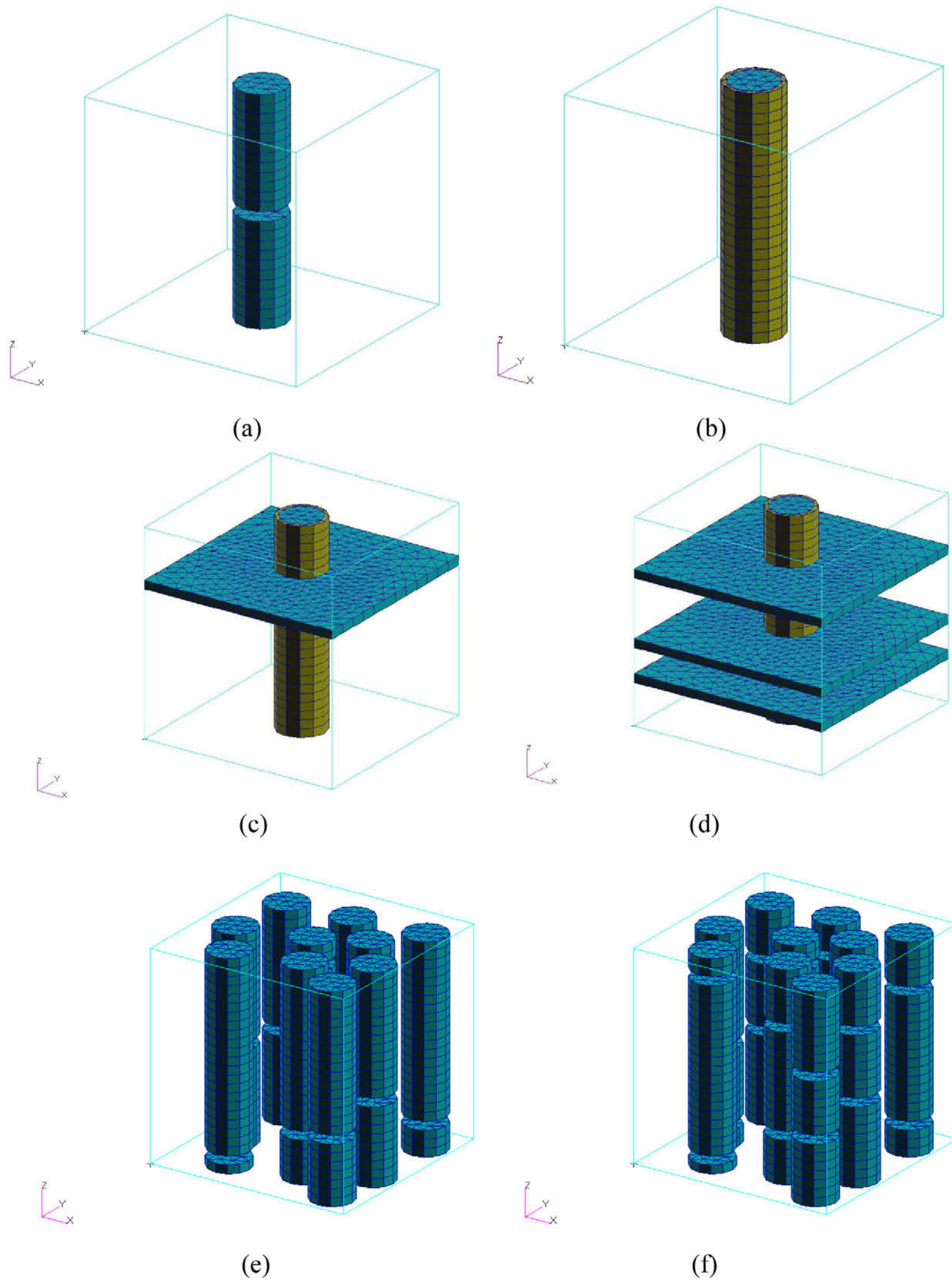

Fig. 1. FE microstructural models used in this study: (a) single fibre model with one damageable layer in fibre, (b) single fibre model with one damageable layer in fibre and fibre/matrix interface layer, (c) single fibre model with one damageable layer in fibre, fibre/matrix interface and a damageable layer in the matrix, (d) single fibre model with one damageable layer in fibre, one fibre/matrix interface and three damageable layers in the matrix, (e) multifibre model with one damageable layer per fibre, (f) multifibre model with two damageable layers per fibre.