Computational Materials Science 111 (2016) 148–156

Contents lists available at ScienceDirect

Computational Materials Science

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

Microscopic damage mechanisms of fibre reinforced composite laminates subjected to low velocity impact

Lei Yang *, Zhanjun Wu, Dongyue Gao, Xin Liu

State Key Laboratory of Structural Analysis for Industrial Equipment, School of Aeronautics and Astronautics, Dalian University of Technology, Dalian 116024, PR China

ARTICLE INFO

Article history:
Received 11 July 2015
Received in revised form 14 September 2015
Accepted 15 September 2015

Keywords:
Fibre reinforced composite
Low velocity impact
Damage mechanisms
Multiscale modelling
Finite element method (FEM)

ABSTRACT

This paper presents an integrated multiscale model for the simulation of fibre reinforced polymeric composite laminate subjected to low velocity impact. The multiscale model is based on the embedded cell method, with detailed microstructure embedded into the macro laminate beneath the impact point, and a transition zone is introduced to link these two scales. Damage model is considered for the fibres and plastic behaviour for the matrix, and cohesive elements are used for the simulation of interface delamination. Both unidirectional and layup embedded cells are considered in the simulation so as to reveal the impact damage mechanisms from monolayer to layup levels. The simulation results indicate matrix cracking is the first damage form which occurs at the bottom of the laminate, and then delamination is induced when the matrix crack propagates to the interface, followed by fibre pull-out and fibre breakage. The simulation results are compared with available experimental results from literatures, with good agreements achieved between them on the damage morphologies. Thus the ability of the presented multiscale model to reveal the damage mechanisms of composite laminate under low velocity impact is validated.

1. Introduction

Low velocity impact is considered as one of the most dangerous damage sources of fibre reinforced polymeric composite structures, as it can cause significant reduction on the stiffness and strength of the structures, but the damage usually cannot be detected from the surface of the component, thus causing great security risks to the structures. For this reason, there have been a great number of researches on the low velocity impact damage of composites by both experimental and numerical approaches in the last decades [1–10]. Recently, with the rapid development of computer power and finite element method (FEM), numerical simulation has gradually become an important method for the research of low velocity impact of composites.

During the low velocity impact process of composite laminates, various damage modes may occur simultaneously, including fibre breakage, matrix cracking, matrix crush and delamination. To take into account all these damage forms, the combination of continuum damage mechanics (CDM) and cohesive zone model (CZM) is often used to simulate the low velocity impact damage of laminates, and has achieved great success in the prediction of damage forms and damage extent of composite laminates subjected to low velocity impact [11]. However, these methods are based on the macro-scale, thus cannot capture the damage mechanisms at the micro-scale, including fibre/matrix debonding, fibre pull-out, and highly localized plastic deformation of the resin. On the other hand, though micro-mechanical simulations can capture these effects, they require high resolution mesh and significant computational power. Therefore, modelling of low velocity impact onto a composite at the micro-scale, capturing all these effects is currently and in the near future not feasible [12]. To overcome this contradiction between simulation accuracy and computational cost, an alternative is to use the multiscale method taking into account the mechanical response at several length scales [13].

The wide range of multiscale methods can be broadly classed into two groups: hierarchical and concurrent multiscale methods. In hierarchical methods [14,15], a “one-way” bottom-up coupling is established where information is passed from lower to higher scales. As a consequence, such methods are efficient in determining the macroscopic stiffness and strength of composites, but may have a limited predictive capability for problems involving damage [16]. Concurrent multiscale methods [17,18] introduce the concept of scale embedding instead of scale transition, thus different scales coexist in adjacent regions of the model. This is especially useful for investigating impact events of composite

* Corresponding author. Tel./fax: +86 411 84706213.
E-mail address: yangl@dlut.edu.cn (L. Yang).

http://dx.doi.org/10.1016/j.commatsci.2015.09.039
0927-0256/

laminates, since large deformations and damage are restricted to a local region. Thus only the damage zone requires a high resolution mesh, so that high simulation accuracy can be achieved with affordable computational cost.

Many researchers [19–22] have carried out multiscale numerical simulation of the damage behaviour of composite materials. However, all these studies were aimed at the quasi-static damage simulation of composites, thus are not applicable to the simulation of low velocity impact. Souza et al. [23] presented a multiscale model for predicting damage evolution in composites due to impact loading. Yet, it was not a concurrent multiscale model as information transfer was required between the global and local scale. Ha-Minh et al. [24] developed a numerical multiscale model for textile woven fabric against ballistic impact, but the scale was limited to the mesoscopic level (yarns). May et al. [12] proposed an adaptive multiscale methodology for modelling composites under high velocity impact. But the microscopic structure was not explicitly included in the model and the damage mechanisms were not discussed. So far, to the best of the authors' knowledge, there is not any research on the multiscale modelling and simulation of fibre reinforced composites subjected to low velocity impact.

In this paper, an integrated multiscale model based on finite element method will be developed for the simulation of fibre reinforced polymeric composite laminates subjected to low velocity impact. The micromechanical damage process of the composites will be discussed and compared with available experimental results, thus to provide a clear understanding of the micromechanical damage mechanisms of fibre reinforced composite laminates under low velocity impact.

2. Multiscale model

In order to simulate the micromechanical damage behaviour of composite laminates under low velocity impact with affordable computational cost, an integrated multiscale model is required that contains both macro boundary conditions and microstructures of the damage zone. As the size of laminate is usually larger than the order of millimeter (length and width $\sim 10^1$–10^2 mm, thickness $\sim 10^0$ mm), while the dimension of constitutes is in the order of micron (fibre diameter $\sim 10^0$ μm), it is impractical to use consistent mesh discretization for the whole model. Thus there should be a proper connection between these two scales. This can be accomplished by the embedded cell model [25], in which the full details of the composite microstructure are resolved in the region of interest, while the remaining part is represented as homogeneous material whose behaviour is given by any suitable homogenization model.

For a composite laminate subjected to low velocity impact, it is well known that the region at the bottom of the laminate just under the impactor is most prone to suffer damage. Therefore, this area is selected as the region of interest where embedded cell with detailed microstructure is inserted, as shown in Fig. 1. The

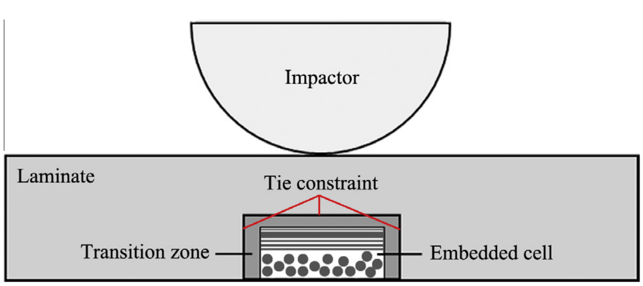

Fig. 1. Schematic illustration of the integrated multiscale model for fibre reinforced composite laminates under low velocity impact.

modelling and simulation platform of this paper is the FEM package ABAQUS/Explicit, and tie constraint [26] provided in ABAQUS is used to accomplish the connection between the macro laminate and the micro embedded cell. However, as the tie constraint is a numerical connection technology, numerical error will be caused at the vicinity of the connecting surface. To avoid this issue, a transition zone is introduced between the macro laminate and the micro embedded cell, which will be further depicted in the following. The impactor is modelled as a hemispherical rigid body with an initial velocity (1 m/s) in the vertical direction. The surface-to-surface contact algorithm is employed to simulate the contact between the impactor and the laminate, using a penalty formulation to take into account the effect of friction.

The size of the laminate is 50 mm × 50 mm × 1 mm, and the four edges of the laminate are fixed. The size of the embedded cell can be determined according to the research requirement and computational power. The transition zone is properly determined by trial computation so that the boundary effect has little influence on the embedded cell. In this paper, both unidirectional and layup embedded cells are used for the simulation. Shown in Fig. 2 is the finite element model of the composite laminate with unidirectional embedded cell. The macro laminate is discretized by eight-node linear brick elements layer by layer, with gradually refined mesh from the edges to the central area. The embedded cell consists of polymeric matrix and randomly distributed fibres whose positions are determined by the RSE algorithm presented by the authors [27]. The transition zone can be divided into two parts: the two ends along the fibre direction contain microstructure of transitional fibres and matrix, with no material damage considered for them but ideal plastic behaviour introduced instead to avoid unrealistic high stress; the rest is treated as equivalent composite material. Matching mesh is used for the transition zone and the embedded cell, while tie constraints are employed between the macro laminate and the transition zone. The size of the embedded cell is 200 μm × 150 μm × 100 μm, the thickness of the transition zone along the x, y and z axis is 20 μm, 25 μm and 10 μm, respectively. The whole model contains 40 fibres with volume fraction of 60% and a total of about 120,000 elements.

Fig. 3 illustrates the finite element model of the composite laminate with layup embedded cell. For simplicity, an orthotropic layup is used here. In order to model the delamination damage, a layer of cohesive elements (2 μm) are inserted between the two plies. The size of the embedded cell is 150 μm × 150 μm × 150 μm, the thickness of the transition zone along the three axes is all 10 μm. The whole model contains about 400,000 elements.

As the element length of the microstructure is quite small, a very small global stable time increment is resulted for the explicit solver. This will result in a very large number of analysis steps, which means unaffordable computational time. One possible solution for this problem is to use the mass scaling technology: by scaling the mass of the elements in the embedded cell and transition zone throughout the simulation, the stable time increment is increased and thus the computation time can be significantly decreased. As the embedded cell and transition zone only account for a very small proportion (<0.01%) of the whole model, increases in mass for these parts during the simulation will have little effect on the overall dynamic response.

3. Material models

The material chosen for the simulation is E-glass/epoxy composite from the World Wide Failure Exercise [28], in which the mechanical and thermal properties for both the laminae and the constitute materials are provided. The mechanical properties of the macro laminate are listed in Table 1, with no damage