Available online at www.sciencedirect.com

ScienceDirect

Procedia Structural Integrity 8 (2018) 33–42

Structural Integrity Procedia

www.elsevier.com/locate/procedia

AIAS 2017 International Conference on Stress Analysis, AIAS 2017, 6-9 September 2017, Pisa, Italy

Evaluation of the effects of the numerical modelling choices on the simulation of a tensile test on CFRP composite

A. Gilioli[a], A. Manes[a]*, M. Giglio[a]

[a]*Politecnico di Milano, via La Masa 1, Milano, Italy*

Abstract

The goal of the present work is to define a method to build a FE model which is able to reproduce an experimental tensile test on CFRP specimen with different stacking sequences (UD and balanced). The defined method assesses the material numerical parameters by means of a simulation that replicates, as a virtual test, the experimental tensile one, and in the future, it will be possible to exploit the data obtained to create a reliable model for the simulation of low velocity impacts. Analyses have been performed using the non-linear solver ABAQUS Explicit. The current work further studies how to model damage and the effect of modifications of the numerical parameters on the results. Indeed, the numerical simulation of composite materials is very sensitive to the numerical choices made. Moreover, from the literature and experiments, the mechanical properties of composites are very variable and hence the evaluation of the model response to such modifications is of particular interest.

Peer-review under responsibility of the Scientific Committee of AIAS 2017 International Conference on Stress Analysis

Keywords: CFRP; FEM; failure; tensile test

1. Introduction and motivations

The topic of numerical simulation of composite materials is widely investigated and many papers are available in the literature. However, such simulations are not straightforward and many research teams continue to work in this field. The present paper focuses on the numerical simulation of a tensile test of a carbon fiber composite CFRP. The goal is to investigate the material properties by means of virtual tests that replicate the actual experiments. Moreover,

* Corresponding author. Tel.: +39-02 2399 8630; fax: +39-02 2399 8263.
E-mail address: andrea.manes@polimi.it

2452-3216
Peer-review under responsibility of the Scientific Committee of AIAS 2017 International Conference on Stress Analysis
10.1016/j.prostr.2017.12.005

the effect of the numerical choices on the numerical models, are also investigated. As far as experimental tests are concerned tensile tests on UD specimens, balanced plain specimen as well as specimens with a central hole have been performed. Finally, the present work is aimed to define a set of data for the material behaviour and reliable methods for the building of numerical models able to reproduce, as a future step, low velocity impacts. Indeed, the simulation of an impact is very time consuming and the initial investigation of a simpler configuration, like a tensile test, is a reasonable choice. Once all the numerical aspects have been properly set-up, a more complex scenario (tensile test on the specimen with hole) is investigated ensuring the goodness of the transferability of the material data and the suitability of technique for the modelling construction. A particular aspect of the research regards the adoption of solid elements for the numerical simulations. In the literature shell, or continuum shell, to model impact on plates have often been the preferred choice, but in recent years, solid elements have been largely used even if the size ratio suggests to use shell elements. The use of solid elements seems to guarantee better results both in terms of numerical values and morphology of damage, Feng (2013), Guo (2013), Boria (2014). The present paper starts with a description of the experimental tests, Section 2, followed by the building of the model, Section 3, including the choices made with regards to how to describe the interaction between each ply and considerations about the mesh. Section 3.3 includes the fine tuning of the fracture properties (of the matrix) in order to better fit the experimental data while always remaining inside the experimental variability. Finally, in the results, Section 4, the accuracy of the simulation obtained by the data calibrated using the experimental data from UD and balanced specimen is shown. The results are then applied in the replication of balanced with hole specimens. This type of specimen allows an assessment of the goodness of the transferability both of the material data and the modelling technique thus can be considered a sort of validation.

2. Experiments

The present activity is focused on the study of pre-impregnated carbon fiber/epoxy unidirectional and multidirectional laminates, and in particular the material used is the MTM45-1/IM7(12K)-145g/m^2-32%RW. It is made of epoxy resin matrix MTM R45-1 32% in weight produced by Cytec and carbon fibers HexTow RIM7 (12K) produced by Hexcel, with an area density of 145g/m^2. The material was provided in form of unidirectional tape, and was subsequently assembled according to the desired stacking sequence and properly cured. The thickness of the lamina is 0.129 ± 0.013mm. The tests were performed according the ASTM D3039 standard with a hydraulic testing machine MTS 810. An extensometer was used to measure the displacement during the tests. The size of the specimen is reported in Table 1.

Table 1. Definition of the specimen

Specimen	layup	Size [mm]	Tabs [mm]
UD	$[0]_8$	250x15x1	56x15x1.5
BAL	$[(0/45/90/-45)_2]_s$	250x25x2	-
BAL with HOLE	$[(0/45/90/-45)_2]_s$	250x36x2	6

3. Specimens modelling

The simulations were performed using the finite element commercial software ABAQUS 6.14 with its dynamic explicit solutor, and three different strategies to model the specimen were studied:

- Perfect Bonding between laminae: PB
- Tie interaction between laminae: TIE
- Cohesive interaction between laminae: COH

All these models involve 3D solid elements with a reduced integration C3D8R, but they differ in the layer modelling techniques. The PB is the simplest model which is used in the preliminary stages to obtain results in a short time. PB refers to a perfect bonding between plies, which means that the laminate is drawn as a single part and then partitioned in order to obtain the laminae to which the specific properties are assigned. All the layers are perfectly bonded (welded)