AN EXPERIMENTAL METHOD AND NUMERICAL SIMULATION FOR COMPOSITE MATERIALS ENERGY ABSORPTION DETERMINATION

F. Garattoni, E. Troiani
MaSTeR Lab, Seconda Facoltà di Ingegneria, Università di Bologna

Keywords: *crashworthiness, composite materials, energy absorption*

Abstract

This paper introduces a comparison between existent test methods and presents the numerical simulation of the one considered the most representative in terms of the energy absorption behavior for composite material. The results of the numerical simulations have been validated through experimental data. This optimized model, obtained with a commercial finite elements code, can provide a simple and economic tool for the designers once extended to a more complex structure.

1 Introduction

Nowadays the growing interest of the aeronautical industry for the use of composite materials in primary aircraft structures requires reliable tools for the design of these structures according to the requirements of occupants safety and crashworthiness.

The overall objective of designing in crashworthiness is to eliminate injuries and fatalities in relatively mild impacts, and to minimize them in severe collisions. This objective can be reached either by modifying the structural geometry of the assembly, or by introducing specific load-limiting devices in the structure to dissipate kinetic energy during a crash [1].

Traditionally, in aircraft, the energy-absorption devices are steel or aluminum structural elements; these materials allow a controlled collapse of the structure during which they absorb energy by folding or hinging, involving extensive plastic deformation. The actual trend of substituting metals with composite materials can improve the energy absorption performances of the devices [2, 3], but it introduces several problems due to the complexity of failure mechanisms that can occur within the material and the combination of fracture mechanisms that lead to structural failure.

Referring to figure 1, few key terms have to be defined in order to understand the energy absorption behavior of a structure [1]:

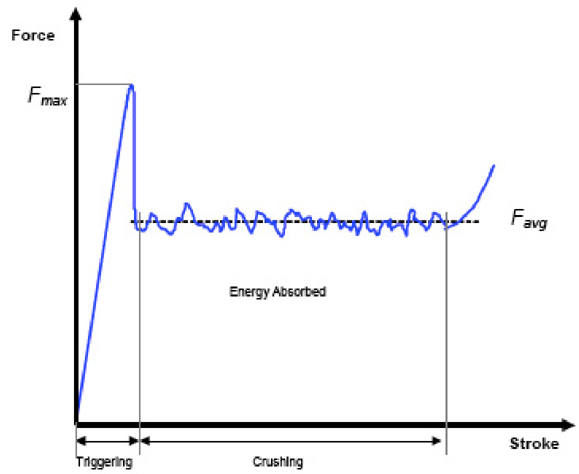

Fig. 1 Exemplar force-stroke history

- stroke, also referred to as crush or displacement, is the amount (length) of structure or material being sacrified during the impact event;

- peak force, also known as maximum load, is the maximum point on the load-stroke or force-displacement diagram;

- average crush force, or sustained load, is the displacement- or time-average value of the crush force;

- crush initiator, also known as trigger mechanism, is a design feature that facilitates the progressive collapse of the structure avoiding the tendency of composite materials to fail in an unpredictable and sometimes unstable manner;

- Energy Absorption (EA) is the total area under the load-stroke diagram

$$EA = \int F dl \tag{1}$$

- Specific Energy Absorption (SEA) is the energy absorbed per unit mass of crushed structure expressed in J/g

$$SEA = \frac{EA}{\rho A l} \tag{2}$$

where ρ is the density of the material, A the cross-sectional area and l the stroke (figure 2).

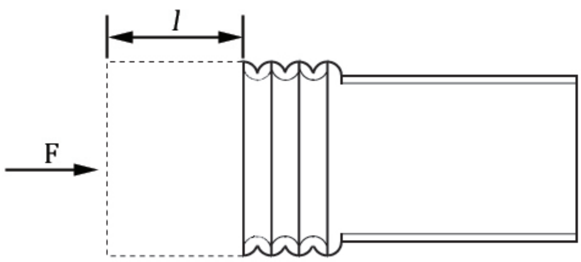

Fig. 2 Representation of a crushed structure

A first attempt to identify the energy absorption related parameters of a crushed structure is the definition of a test method for the determination of the SEA.

In literature, it is possible to find only a limited number of attempts of coupon size test methods for composite materials. They can be divided into two main categories based on the kind of specimen they use:

- flat specimen tests [4, 5, 6];

- self-supporting specimen tests, usually thin walled tubes [7, 8].

The flat specimen test method consists generically in a flat coupon crushed in between two anti-buckling supports as it can be seen in both the fixtures developed by the Army Research Laboratory (ARL) [4], figure 3, and University of London [6]. The flat specimen method has the

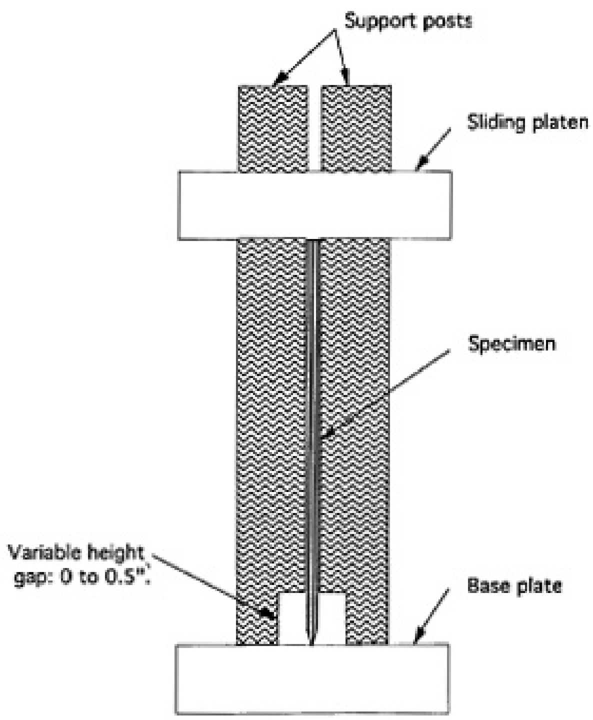

Fig. 3 Sideview of the baseline model of fixtures developed by ARL [4]

advantage of an easy manufacture of the coupons, while the production of tubes [7, 8, 9] for the self-supporting specimen is a more difficult and more expensive process. The two test methodologies differ in terms of main fracture mechanisms: the flat specimen crashes principally through lamina bending or splaying mode, which is characterized by two fronds and a long central intralaminar crack (fig. 4), while in the self-supporting specimen (fig. 5) the fracture mechanism involves a combination of fracture modes such as splaying (fronds formation) and fragmentation. The analysis of the energy absorption capability reflects the difference between the fracture mechanisms of the two kinds of specimens: in the flat specimen the main absorbing mechanism is matrix crack growth (low energy absorbed), while in the self-supporting specimen, besides the intralaminar crack growth, the fracture of lamina bundles occurs, absorbing a higher amount of energy. In figure 6, the SEA of a generic flat test (dashed line) is compared to the one produced by the crash of a self-supporting specimen (plain line): the flat specimen does not achieve a sustained crush that leads to an adequate level of energy absorbed [10]. The better reliability of the self-supporting specimen test method compared to the flat specimen tests depends also on the absence of external factors introduced through the fixture, necessary to sustain the flat specimens, such as the knife-edges height (or the unsupported length of the coupon, figure 3) or the closing force of the anti-buckling supports (that depends uniquely from the test operator) [11]. In figure 7 the comparison between the specific energy absorption of flat specimen in function of the knife-edges height and the SEA obtained with self-supporting specimen is shown.

Fig. 4 Crushed flat specimen

Fig. 5 Crushed self-supporting specimen

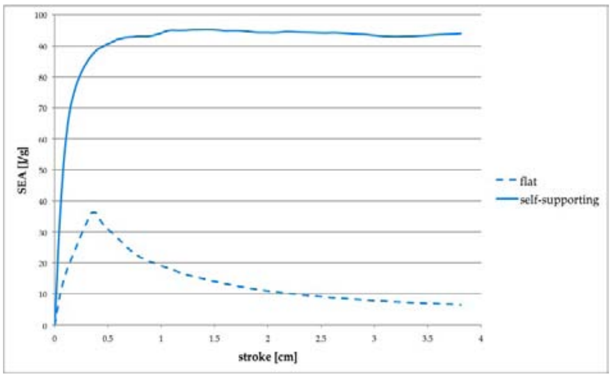

Fig. 6 Typical SEA behavior of flat and self-supporting specimen tests

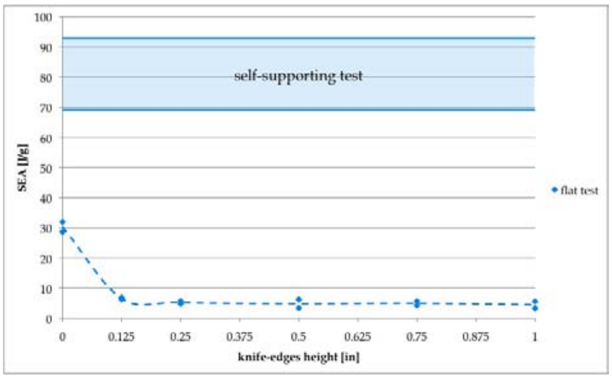

Fig. 7 SEA comparison of flat and self-supporting specimen referred to flat coupons unsupported height

This work focuses on the definition of a test procedure using self-supported coupons with a periodic semi-circular shape; the data obtained from the experiments are used as a comparison to validate a numerical model of progressive damage for a composite material included in the finite element analysis (FEA) software Abaqus/Explicit. The numerical model presented is not intended to simulate every single fracture mode occurring in the real test specimen, but the macroscopic behavior of the crash. Therefore, its purpose is to be a quick and economic tool to give a first estimation of the energy absorption capability of a structure once the numerical behavior of the test simulation concords with the real test.

2 Methodology

2.1 Experimental test procedure

The coupon size test with self-supporting specimen presented in this section captures the advantages of both the two test philosophies: in fact, the particular specimen shape (fig. 8) does not require the support of any anti-buckilng fixture and at the same time can be easily produced in a dedicated mold (fig. 9) as a corrugated plate. The mold is designed for the use with prepreg material systems.

The corrugated plate is made pressing a stack of plies of prepreg between the two molds and then cured in autoclave according to the indications provided by the supplier. The mold is made to host 12 plies of unidirectional prepreg (average thickness 0.15 mm) or 8 plies of plain weave material (nominal thickness 0.23 mm). Once machined, each plate is about 210 mm x 180 mm and 1.8 mm thick, from which is possible to cut six specimens, four with three repetitions of the half-circular modulus (60 mm x 80 mm) and two with five repetitions (80 mm x 80 mm). Each specimen is triggered with a 45° single side chamfer, act to reduce the initial peak force. The specimens produced with unidirectional tape have a $[0/90]_{3s}$ layup, while the ones made in plain weave fabric are an eight plies stacks.

Specimens are tested in vertical configuration, compressed between two steel plates that slide along four steel shafts with self-aligning ball bearings (fig. 10) at a quasi-static speed

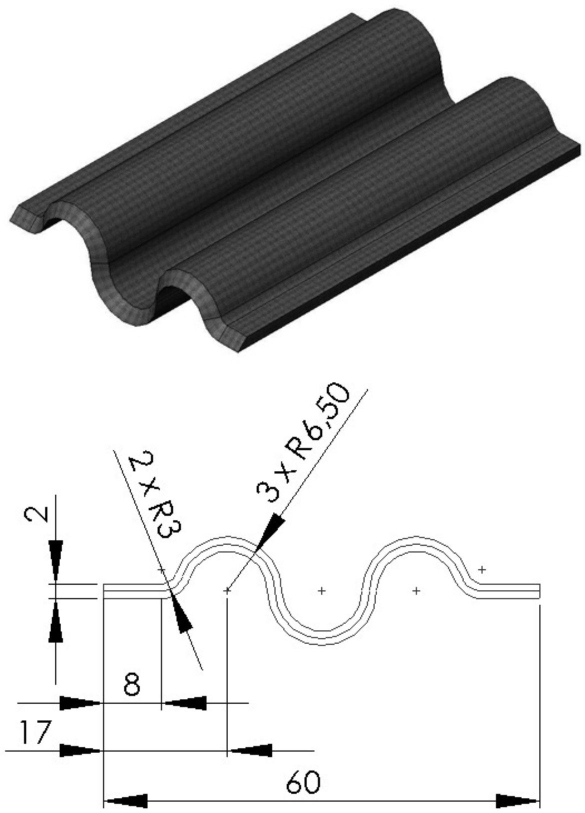

Fig. 8 Example of self-supporting specimen: model and draw (dimensions in mm)

Fig. 9 Mold for self-supporting specimen with carbon fiber plate

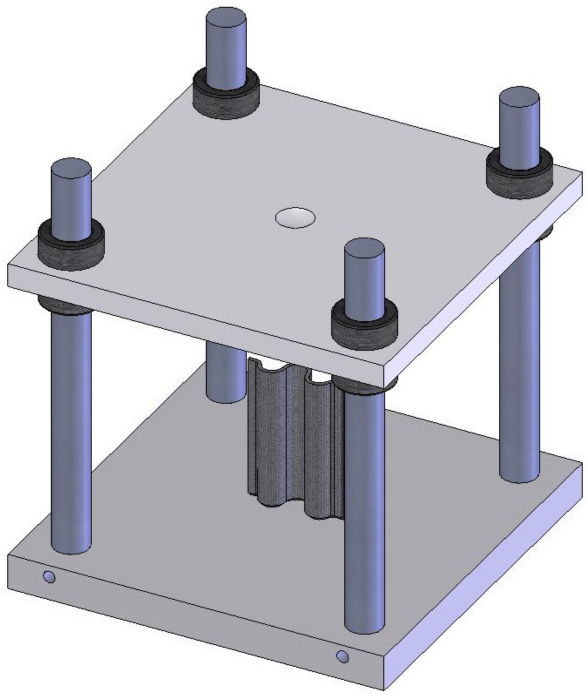

Fig. 10 Test configuration

rate of 50 mm/min. A displacement of half the height of the specimen is imposed. The choice of a quasi-static speed rate test over a dynamic crash test is governed by the higher expenses and the more complicated test equipment the latter requires. Performing the tests at a quasi-static rate does not influence the test results though: even if there is a lack of consensus about the influence of test speed on the energy absorption in literature, a side test campaign confirmed that it does not affect the SEA.

For each specimen are recorded the load and the movable head displacement of the hydraulic press where the test is performed. From the force-displacement history, applying (1) and (2) is possible to calculate the SEA of the tested coupon; a typical plot of load, EA and SEA as function of the stroke is shown in figure 11.

The materials tested are all carbon fiber reinforced plastics (CFRP) with epoxy resin systems, both unidirectional prepreg tapes and plain weave prepreg fabrics, and the average values of SEA obtained from different tests can vary from 68 J/g to 93 J/g according to different material properties.

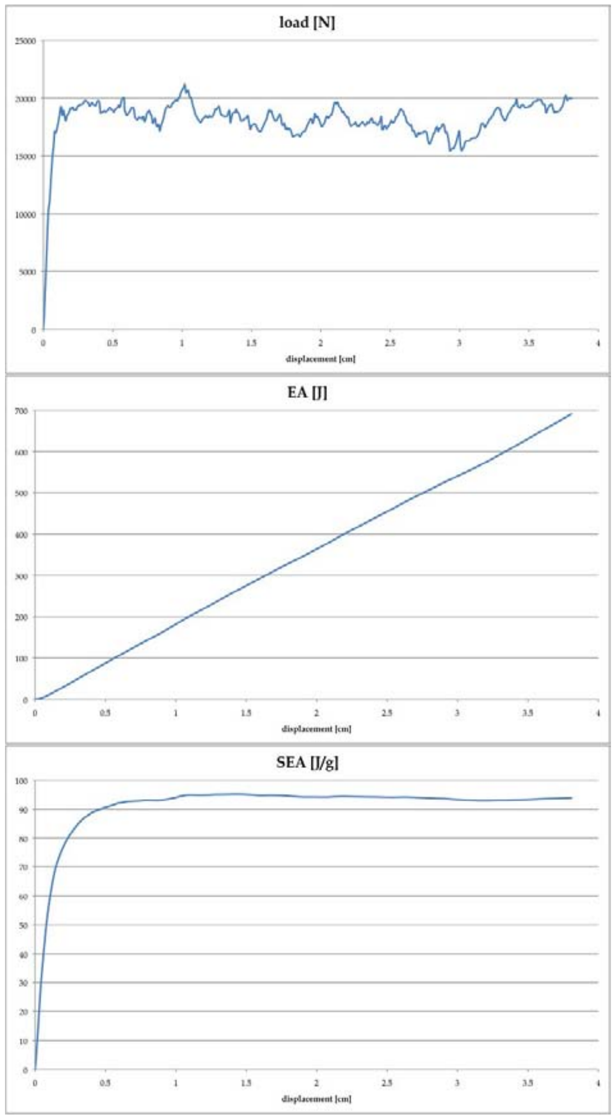

Fig. 11 Typical test plots (from top to bottom): load-displacement, EA-displacement, SEA-displacement

2.2 Numerical model

The numerical model simulates the test in its whole configuration, that means that not only the specimen is represented but also the two steel plates (fig. 12). The two steel plates are modeled as rigid bodies, so they do not interfere with the specimen crush, but their presence simplifies the definitions of boundary conditions and loads on the specimen.

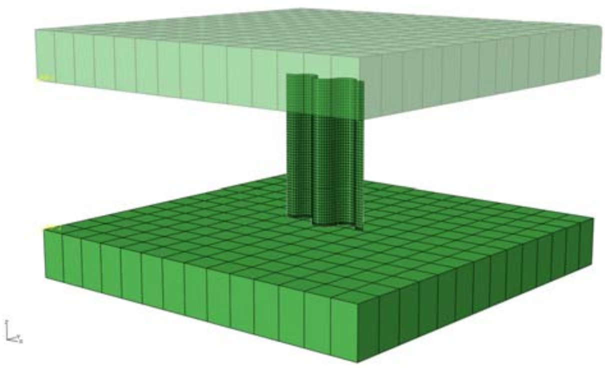

Fig. 12 Model in Abaqus/Explicit environment

The specimen is imported in Abaqus environment as a simple surface and then modeled with S4R elements (4-node, doubly curved shell with reduced integration, hourglass control and finite membrane strains [12]). The stratification of composite material is modeled using the composite layup feature inside Abaqus/CAE, that allows to define for each set of selected elements the number of plies, material properties, ply thickness and fiber orientation; the definition of the transverse shear of the section is also required. For this study, the specimens consist of four different elements set with four different layups: a $[0/90]_{3s}$ layup constitutes the main section (fig. 13), then three rows of elements for a total height

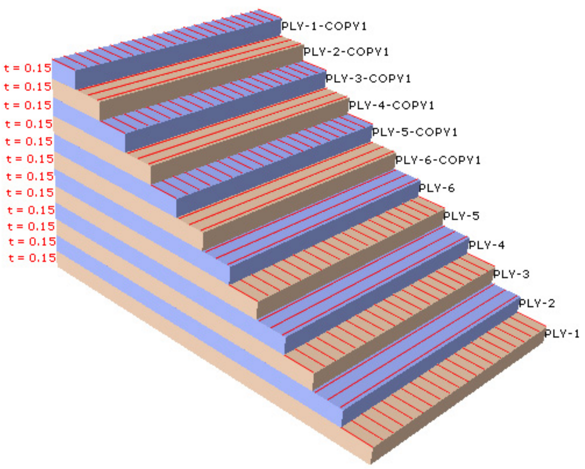

Fig. 13 $[0/90]_{3s}$ layup of the specimen (full section)

equal to the real thickness are modeled with a decreasing number of plies to simulate the trigger (fig. 14).

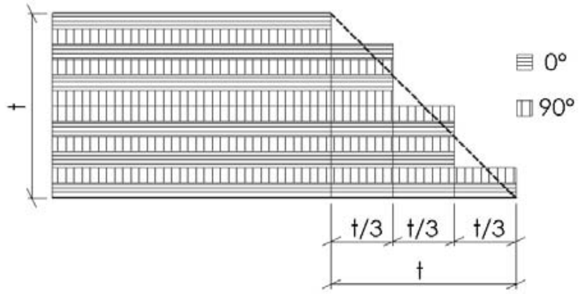

Fig. 14 Trigger model

The progressive damage model used in this analysis [13] needs the definition of several properties for the material, some of them obtained using standard tests and others obtained with experimental procedures [14]. The elastic properties of the tested material (CFRP, unidirectional tape) and its ultimate strengths are shown in table 1 and 2 respectively; the material density is 1.55 g/cm^3 while the nominal ply thickness is 0.15 mm.

E_1 [GPa]	E_2 [GPa]	ν_{12}	G_{12} [GPa]
126.86	8.41	0.309	
E_{1c} [GPa]	E_{2c} [GPa]	ν_{12c}	4.205
113.76	10.135	0.309	

Table 1 Elastic properties for a carbon fiber reinforced epoxy

σ_{u1t} [MPa]	σ_{u2t} [MPa]	σ_{u12} [MPa]
2199	50.263	
σ_{u1c} [MPa]	σ_{u2c} [MPa]	154.443
1469	198.569	

Table 2 Ultimate strength for a carbon fiber reinforced epoxy

The fabric reinforced ply is modeled as a homogeneous orthotropic elastic-plastic material whose properties are degraded due to fiber/matrix cracking. The properties degradation is realized through the introduction of damage parameters into the elastic compliance matrix $\boldsymbol{S}$ in its general form:

$$\boldsymbol{S} = \begin{pmatrix} \frac{1}{E_1(1-d_1)} & -\frac{\nu_{12}}{E_1} & 0 \\ -\frac{\nu_{12}}{E_1} & \frac{1}{E_2(1-d_2)} & 0 \\ 0 & 0 & \frac{1}{G_{12}(1-d_{12})} \end{pmatrix} \quad (3)$$

The three scalar parameters d_1, d_2 and d_{12} represent the damage evolution: d_1 and d_2 are associated to fiber damage or failure, while d_{12} is related to matrix micro-cracking due to shear deformation. The damage parameters have value $0 \leq d_i \leq 1$ and are monotonically increasing quantities. The evolution of damage variables is a function of a corresponding damage threshold and a fracture energy per unit area.

According to the previous consideration about the influence of the speed on the energy absorption, in the numerical simulation a speed rate of 0.83 mm/sec is imposed with a time step of 45 sec.

The deformed shape of the specimen after the whole analysis is performed is shown in figure 15. As for the experimental test, a force-

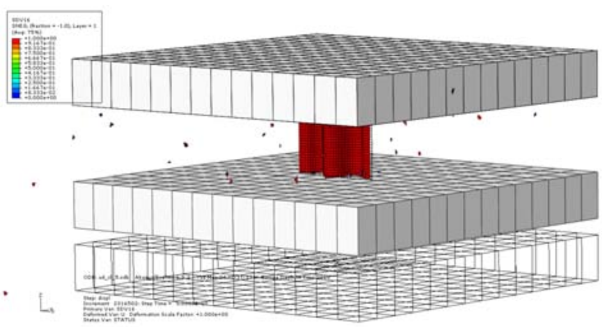

Fig. 15 Numerical simulation: undeformed and deformed shape (red = active element; blue = dead element)

displacement history is obtained, referring to the reaction force in the displacement direction on the top plate and considering the bottom plate displacement. From the data set obtained from the simulation, applying (1) and (2) is possible to obtain the evolution of SEA in function of the bottom plate displacement as shown in figure 16.

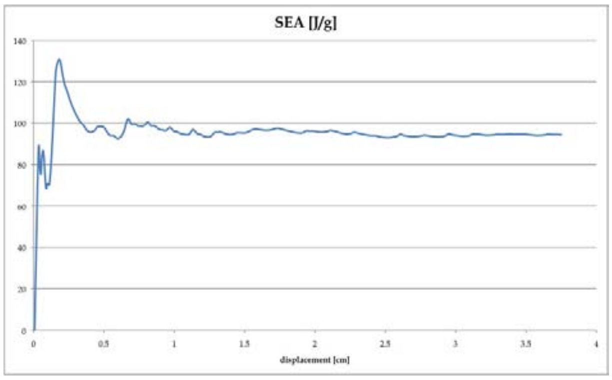

Fig. 16 SEA vs. displacement plot obtained from the numerical simulation results

3 Results and discussion

Two aspects of the numerical model results are here discussed: the differences between the deformed shape of the virtual specimen and the real crushed coupon and the comparison of the energy absorption behaviors of the two.

The main aspect that can be noted from the comparison between the real crushed specimen (fig. 5) and the numerical specimen (fig. 15) is the absence of the two fronds in the latter. The way the specimen is simplified and modeled in Abaqus environment in fact does not take into account the splaying mode and therefore the central intralaminar crack that forms in the reality. This factor does not affect the results in terms of energy absorption capability: the formation and propagation of the intralaminar crack involves mainly the material matrix, which contribution to the dissipated energy in the case of an opening mode loading is negligible. More accurate results in terms of simulation of the deformed shape can be obtained by modeling each ply of the specimen as a single surface and then assigning cohesive elements at their interface, but this will increase the time of model definition and analysis.

The comparison between the SEA vs. displacement plots obtained with the experimental tests (average value) and the numerical simulation is shown in figure 17, where the plain line is the result of Abaqus analysis and the dashed line is the experimental result.

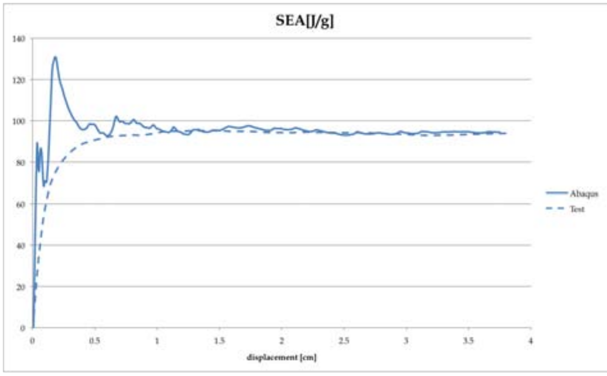

Fig. 17 Comparison between average experimental data and numerical simulation results

The result obtained with Abaqus differs from the real tests in few aspects, but at a first look it can be said that the overall behavior is similar and comparable.

The most remarkable difference of the numerical simulation from the experimental test is the presence of three peaks at the beginning of the crush. These three peaks represent the transient state, in fact the higher one corresponds exactly to a displacement value of 1.8 mm, the specimen thickness. The presence of these peaks can be explained with the coarse discretization of the trigger (fig. 14): while in the real test the trigger (the dashed line in the figure) crashes progressively, in the numerical model there are three well identified different sections with relatively different behavior to the load applied.

It can be noted that the SEA of the numerical model, after the end of the transient state, is not perfectly flat and overlapped to the real test but oscillates around the test values and once again it can be explained with the discretization of the specimen surface and the failure of the single finite element.

Excluding the transient, the average SEA obtained with Abaqus is 94.680 J/g while the average SEA on the same portion of crushed specimen of the real test is 94.175 J/g, with an error of 0.53%.

4 Conclusions

In this paper an experimental test procedure for the determination of the specific energy absorption of composite material is presented, as well as a numerical model of progressive damage for a fiber reinforced material.

The results obtained from the experimental campaign are used as a comparison to validate the behavior of the numerical model implemented in Abaqus/Explicit.

The objective of the numerical simulation is the macroscopic representation of the behavior of the structure in terms of energy absorption, therefore the single fracture modes that occur in the reality are not intended to be modeled.

The main purpose of this numerical model is to be a quick and economic tool to estimate the SEA of a complex structure performing only few experimental coupon size tests of the material the structure is made.

The comparison between the results obtained from the experimental tests and the numerical simulations shows a good agreement between the relative energy absorptions, that encourages further investigations on different specimen geometries.

5 Acknowledgments

The authors wish to thank Gianluca Molinari for the big support in all the Abaqus related matters and Paolo Proli for both the realization of the mold and the technical support.

Copyright Statement

References

[1] VV. AA. Chapter 14 - crashworthiness and energy management. *MIL-HDBK-17-3F*, 3:1–10, Dec 2005.

[2] George C. Jacob, John F. Fellers, Srdan Simunovic, and J. Michael Starbuck. Energy absorption in polymer composites for automotive crashworthiness. *Journal of Composite Materials*, 36(07):813–850, Mar 2002.

[3] Joe J. Carruthers, A. P. Kettle, and A. M. Robinson. Energy absorption capability and crashworthiness of a composite material - a review. *Applied Mechanics Review*, 51(10):635–649, May 1998.

[4] J. André Lavoie and John Morton. Design and application of a quasistatic crush test fixture for investigating scale effects in energy absorbing composite plates. *NASA Technical Publications*, CR 4526:1–68, Jun 1993.

[5] J. André Lavoie and S. Kellas. Dynamic crush tests of energy-absorbing laminated composingte plates. *Composites: Part A*, 27(6):467 – 475, Sep 1996.

[6] Silvano Cauchi Savona and Paul J. Hogg. Investigation of plate geometry on the crushing of flat composite plates. *Composites Science and Technology*, pages 1–12, Dec 2005.

[7] Dieter Kohlgrüber and Argiris Kamoulakos. Validation of numerical simulation of composite helicopter subfloor structures under crush loading. *Proceedings of the 54th AHS Annual Forum*, Washington, DC, May 20-22:340–349, Mar 1998.

[8] Gary L. Farley and Robert M. Jones. Energy-absorption capability of composite tube and beams. *NASA Technical Publications*, TM 101634:1–248, Jun 1989.

[9] Thomas J. Brimhall. Dynamic energy absorption modes of braided carbon/vinyl ester composite crush tubes. pages 1–18, Aug 2006.

[10] Paolo Feraboli and Francesca Garattoni. Development of a test method for composite materials energy absorption: corrugated specimens. *Proceedings 48th AIAA Conference*, Honolulu, HI April 23-26:1–19, May 2007.

[11] Paolo Feraboli, Francesca Garattoni, and Francesco Deleo. Efforts in the standardization of composite materials crashworthiness energy absorption. *Proceedings of the 22nd ASC Technical Conference*, Seattle, WA Sept 17-20:1–19, Aug 2007.

[12] Dassault Systèmes. Abaqus 6.9 documentation. Jul 2009.

[13] Alastair F. Johnson. Modelling fabric reinforced composites under impact loads. *Composites: Part A*, 32:1197–1206, Aug 2001.

[14] Silvestre T. Pinho, Paul Robinson, and Lorenzo Iannucci. Fracture toughness of the tensile and compressive fibre failure modes in laminated composites. *Composites Science and Technology*, 66:2069–2079, Jul 2006.