Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 167 (2016) 80 – 87

Procedia Engineering

www.elsevier.com/locate/procedia

Comitato Organizzatore del Convegno Internazionale DRaF 2016, c/o Dipartimento di Ing. Chimica, dei Materiali e della Prod.ne Ind.le

Numerical, Experimental and Analytical Correlation for Predicting the Structural Behavior of Composite Structures Under Impact

F.Di Caprio[a]*, A. Langella[b], V. Lopresto[b], G. Caprino[b]

[a] *CIRA – Italian Aerospace Research Centre, Capua, Italy*
[b] *University of Naples "Federico II" Department of Industrial Engineering - Aerospace Branch, Italy*

Abstract

In the present work, numerical, experimental and analytical results regarding impact events on composite structure are presented. The test case consists in a classic 24 plies CAI specimen (100×150 mm) subjected to 10 J impact. The work can be divided into two phases. The first phase is finalized to the definition of a procedure able to provide a robust numerical model, which can simulate accurately the structural response of composite plates subjected to impact events. At this phase, the numerical results are compared with analytical ones. In the second phase, both inter- and intra-lamina failure are considered. Regarding the inter-laminar failure, an experimental-numerical procedure is defined in order to set the right parameters related to cohesive behaviour. For both phases, trade-off analyses on the main numerical parameters are performed. All numerical results are compared with experimental ones in terms of both energy balance and damaged area.

Peer-review under responsibility of the Organizing Committee of DRaF2016

Keywords: Composite Material, Impact analysis, Finite elements analysis, Experimental tests

1. Introduction

Due to their high specific mechanical properties, fiber reinforced composites are widely used in aerospace applications. The high weight-specific stiffness and strength of composite materials allows for a significant

* Corresponding author. Tel.: +39 0823 623538; fax: +39 0823 623335.
E-mail address: f.dicaprio@cira.it

Peer-review under responsibility of the Organizing Committee of DRaF2016
doi:10.1016/j.proeng.2016.11.672

reduction of airframe weight. However, despite their excellent in-plane properties, the poor transverse strength of laminated composites makes them prone to damage induced by transverse loading conditions. Therefore, impact loads can cause substantial damage in composite structures, causing a significant reduction of strength and stiffness [1]. Low energy impacts can trigger extensive damage to the fibers and/or to the matrix [2,3], leading to a substantial loss in post-impact strength [4].

Fiber-reinforced polymer composites, especially CFRP, are very susceptible to reductions in strength due to accidental impact damage. In the case of structures subject to compression load, such reduction in strength can be significant and it is typically taken into account in the design phase by adopting conservative material design allowable. For such a reason, it is very important to develop methodologies able to provide useful information regarding the actual damage state of the structure, estimating the capabilities of the component to sustain the operative load.

In the present work, a numerical, experimental, and analytical correlation regarding the impact event on composite structure is presented. The test case is a classic 24 plies CAI specimen (100×150 mm) with a stacking sequence of $[45,-45,0,0,90,0]_{2s}$ subjected to impact at 10J. The two test cases were used to correlate the numerical results with the analytical and the experimental ones. Since the impact at 10 J did not generate significate failure, the specimen subjected to the 10 J impact was used to compare the numerical results to the experimental ones and to the analytical solution, considering a linear elastic behavior of the material. In a first phase different numerical models were considered, assuming different discretizing levels and different element formulations. In particular, the contact algorithm used both for contact between impactor and plate, and between plate and fixture (a rigid support as prescribed by the normative) was investigate. Under this assumptions, a robust numerical model able to simulate accurately the structural response of composite plates subjects to impact event that do not generate failure can be defined. The last can be considered as starting point for more complex analyses. The best model was used in the second phase taking into account both inter-lamina [5-7] and intra-lamina [7,8] failure. The delamination was simulated through a contact surface with cohesive behavior and failure option placed between each sub-laminate [9]. An experimental-numerical procedure was used to define the right parameters related to cohesive surface. Adequate failure criteria were defined for each ply in order to simulate the failure of fiber and matrix (Hashin criteria [10]). A trade-off on all numerical parameters that influence this kind of analysis was performed. All numerical results are compared with experimental ones in terms of both energy balance and damaged area.

2. Numerical model definition

The analyses described are characterized by a huge number of numerical parameters, that can be set assuming a step-by-step methodology.

The first phase consists in the definition of a robust numerical model able to simulate the impact events on composite structure without considering any failure. Hence, only the material elastic behavior has been considered. In such a way, the numerical parameters can be easily set. Several numerical models were defined with increasing complexity level and with different setting of numerical parameters (element size, element formulation, etc.). In this context, for sake of brevity, only the main results are presented. The numerical results, relating to this first phase, were compared with analytical ones. All that, since the experimental tests produced some failure even at 10 J.

In the second phase, after setting the main numerical parameters, inter- and intra-lamina failure option were added increasing the complexity level, but increasing at the same time also the accuracy level.

2.1. Analytical results

Analytical solutions are available in literature for particular cases like rectangular composite plate with simple supported along the edge. The motion of an orthotropic plate is governed by the equation:

$$D_{11}\frac{\partial^4 w}{\partial x^4} + 2(D_{12} + D_{66})\frac{\partial^4 w}{\partial x^2 \partial y^2} + D_{22}\frac{\partial^4 w}{\partial y^4} + D_{16}\frac{\partial^4 w}{\partial x^3 \partial y} + D_{11}\frac{\partial^4 w}{\partial x \partial y^3} + I_1 \ddot{w} = 0 \quad (1)$$

Considering the boundary condition of a simple supported plate, the previous equation is satisfied when the displacements are expanded into the double series:

$$w(x,y,t) = \sum_{m,n=1}^{\infty} \alpha_{mn} \sin\frac{m\pi x}{a} \cos\frac{n\pi y}{b} \tag{2}$$

Substituting Eq. (2) into Eq. (1), and considering the expression of the applied load, it is possible to obtain for each mode:

$$\ddot{\alpha}_{mn} + \omega_{mn}^2 \alpha_{mn} = \frac{4F}{abI_1} \sin\frac{m\pi x_0}{a} \sin\frac{m\pi y_0}{b} \tag{3}$$

where ω_{mn} is the natural frequency and it is given by:

$$\omega_{mn}^2 = \frac{\pi^4}{a^4 I_1}\left[D_{11}m^4 + 2(D_{12} + 2D_{66})m^2n^2r^2 - 4D_{16}m^3nr - 4D_{26}mn^3r + D_{22}n^4r^4\right]. \tag{4}$$

If m and n vary respectively from *1* to p and *1* to q, the motion of the plate is described by $N = p \cdot q$ equations. In order to completely define the system, the equation that describe the motion of impactor and the initial condition must be taken into account too. It is possible to note that in this case all terms of D matrix were considered. Different modes were considered, but beyond $m = n = 5$, no significant differences can be appreciated. Figure 1 shows the contact force and the impactor displacement as a function of time. It is possible to note that the solution converges increasing the number of modes.

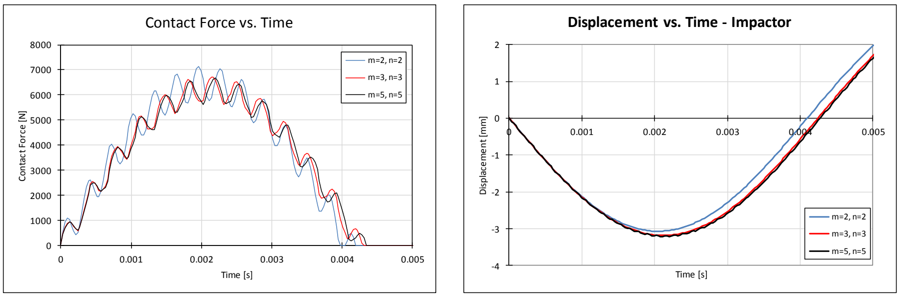

Fig. 1. Analytical solution: Contact force (left side) and impactor displacement (right side)

2.2. Numerical results

The first set of numerical results regards the element formulation and the discretization level. In particular, three different element formulations were investigated: shell, tick shell (TS, discretized by means of continuum shell - CS) and solid (SD). Moreover, five discretization level through the thickness (1, 2, 4, 12, and 24 elements), and three in plane discretization level (2, 1, and 0.5 mm) were taken into account.

In this context, only the main results are reported. In next figures, the contact force and impactor displacement for all element formulation e for all discretization level through the thickness are reported. The graphs for each in plane mesh size are not reported, since they do not provide further significant information. For each numerical results, also the analytical solution is reported. All results have been obtained considering an in plane mesh size equal to *1 mm*. From Figure 2 and Figure 3, it is possible to note that assuming different element formulation and different number of element thought the thickness could lead to estimate the stiffness of the plate mistakenly. Using few elements through the thickness or considering a shell model leads to an overestimation of the stiffness, therefore the contact duration in smaller and the maximum contact force in higher than the analytical solution. Despite using a greater number of elements through the thickness can lead to increase a lot the computational time. Modelling the plate with 24 solid elements, or 12 tick shell elements through the thickness provide a very good results, so this kind of model have been used for the following analyses.

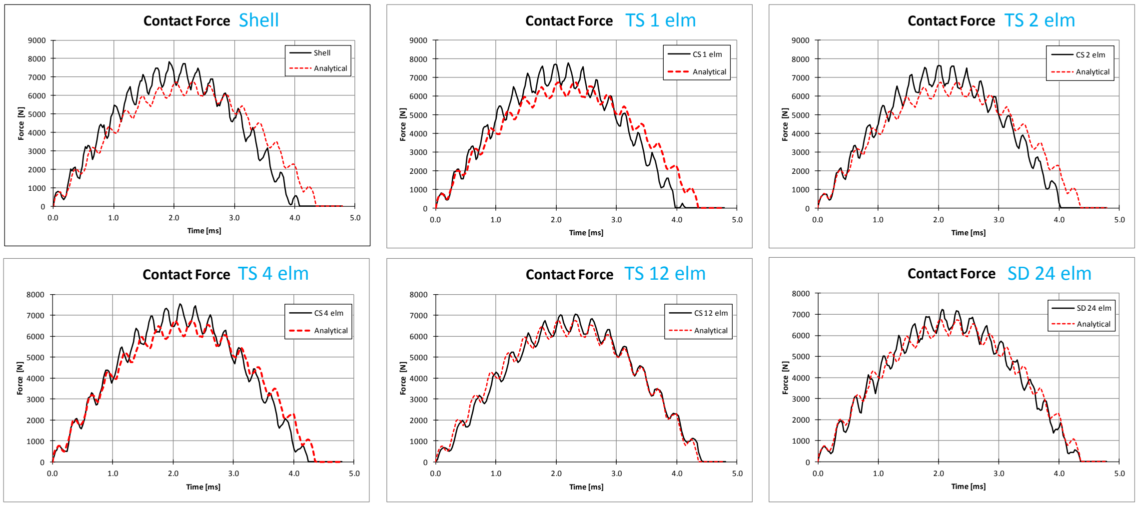

Fig. 2. Analytical-Numerical Correlation: Contact force for different element formulation e different number of element through the thickness

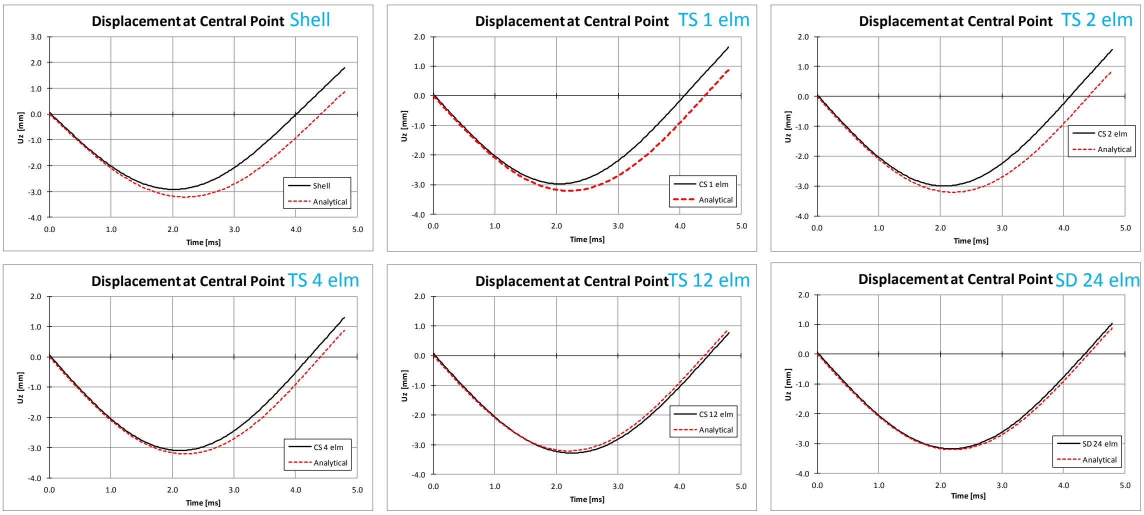

Fig. 3. Analytical-Numerical Correlation: Impactor displacement for different element formulation e different number of element through the thickness

3. Setting of cohesive parameters

In order to increase the complexity level of the model and to obtain a more realistic simulation, it is mandatory to simulate also the possible inter lamina failure that could occur during the impact event. A built-in PFA formulation, based on the Hashin criteria for the failure onset and an instantaneous degradation rules for the material stiffness moduli, has been used to simulate the intra laminar failure.

The cohesive model adopted make use of a classical traction-separation law. Therefore, three type of parameters have to been set: initial stiffness (in normal and shear direction – K_{nn} and K_{ss}), the maximum stress (traction and shear – N_{max} and S_{max}) and the failure energy (G_I and G_{II}).

In this work, three different experimental test have been used to set these 6 parameters: three-point bending, double cantilever beam (DCB) and end-notch flexural (ENF) test.

The first one has been used to set the correct value of the stiffness in normal and shear direction. The specimen is a *100 mm* long beam, characterized by a width and thick both equals to *4.464 mm*. A layup identical to the one of the principal test case has been considered. A numerical model with 24 elements and 23 interface through the thickness has been defined. The elements' in plane size is equal to *1 mm*. The best stiffness values are the minimum values that guarantee the minimum penetration and sliding between each ply, and a good correlation with experimental results in term of load-displacement curve (global stiffness).

As a starting point, the stiffness values have been set equal to the material properties ($K_n = E_2$ and $K_s = G_{12}$). In Figure 4 and Figure 5, the main results as function of respectively normal and shear stiffness scale factor are reported.

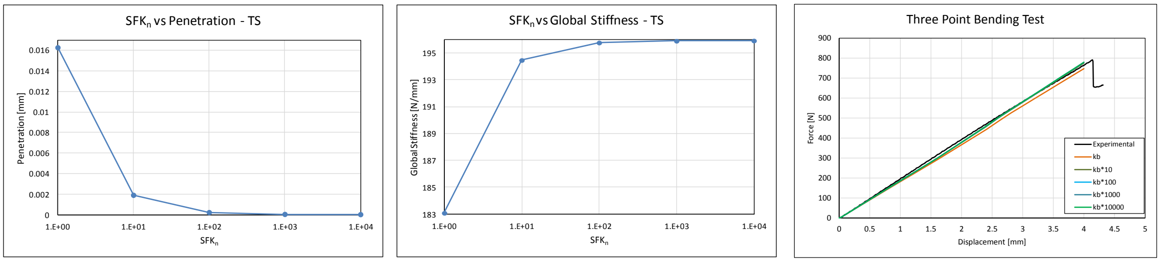

Fig. 4. Penetration vs. Normal Stiffness scale factor and global stiffness vs. Normal Stiffness scale factor

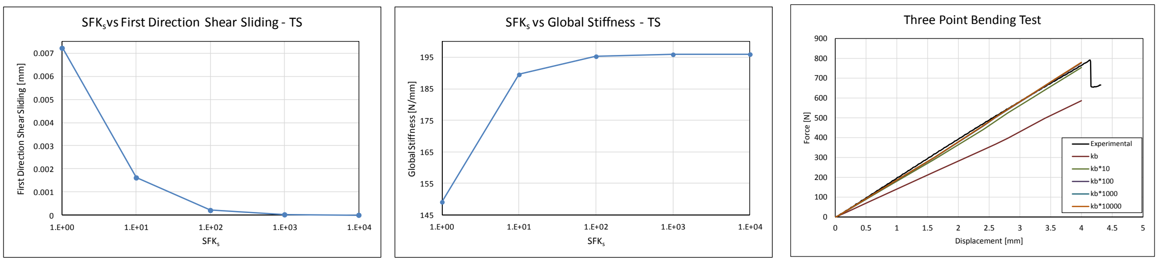

Fig. 5. Sliding vs. Shear Stiffness scale factor and global stiffness vs. Shear Stiffness scale factor

According to Figure 4 and Figure 5, it is possible to note that an increase of the scale factor led to a decrees of both penetration and sliding, and to an increase of the global stiffness which reaches the experimental data. Using too high values could lead to overestimate the fracture energies, since high values of the stiffness are related to smaller values of the strain energy, up to failure onset (the maximum stress is fixed).

The double cantilever beam and the end notch flexural test have been used to set the correct values of maximum stress respectively in the normal and in the shear direction. Furthermore, the tests provide respectively G_I and G_{II} fracture energy values.

Other two numerical models have been defined, in order to simulate the experimental tests of DCB and ENF. The specimen dimensions are provided by related standard (ASTM 5528 and EN 6034). Also in this case, the numerical models have an in plane element size equal to *1 mm*. The stiffness obtained by the previous tests has been assumed. Moreover, the maximum normal and shear stresses have been changed in order to fit with the experimental curve to the best. In Figure 6, the DCB and ENF numerical and experimental results are reported. For sake of clarity, only few results have been reported. For this set of analyses, the damage evolution has not been considered. Therefore, a sudden failure occurred after the damage onset. It is possible to note that it is quite easy to obtain the right value with few numerical analyses. Since some data are classified reserved, the results in this section are provide in no-dimensional form.

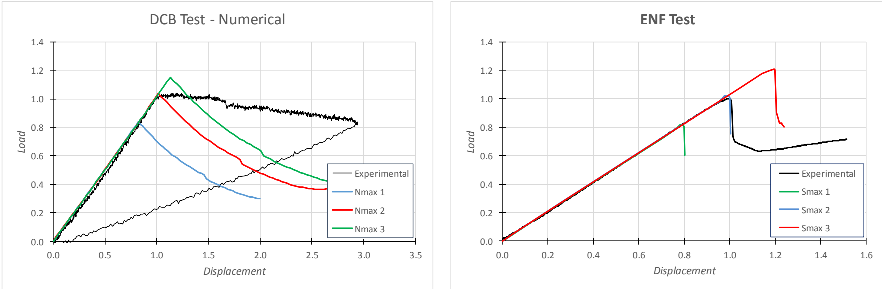

Fig. 6. Setting of Maximum normal stress (left side) and shear stress (right side)

Finally, the damage evolution, with and without viscous regularization, has been activated.

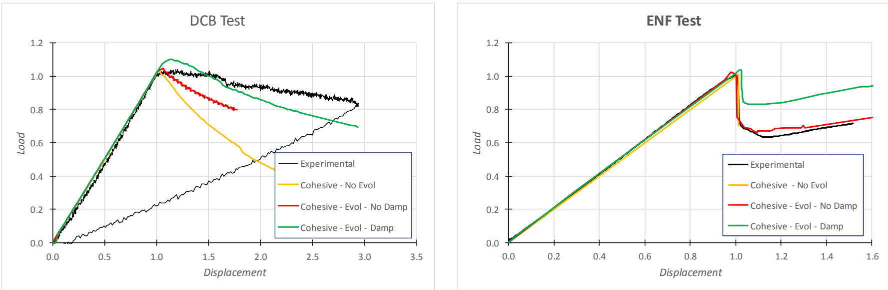

Fig. 7. Setting of damage evolution for DCB (left side) and ENF (right side)

Unfortunately, the damage evolution in the DCB test does not reach a good numerical-experimental correlation. On the contrary, a good correlation has been obtained for the ENF test. The reason could be addressed to the damage evolution rules implemented in the finite element code, that could be not adequate to follow a slow damage propagation.

4. Numerical-Experimental correlation

The results obtained by the two previous procedures have been used to model a detailed composite plate with impactor, pin, and fixture. The impactor, the fixture, and the pins were considered rigid with a fine mesh (impactor = *0.5 mm*; fixture = *1 mm*), while the plate has been discretized with 12 and 24 elements through the thickness, and with an in-plane mesh size of about *2 mm*. In Figure 8, the geometrical model and a generic frame of the simulation are reported. A contact interaction has been set between the plate and the fixture, between the plate and the impactor, and between the plate and the pins. A cohesive interface has been placed between each sub-laminate of the plate, leading to a total of 11 and 23 cohesive interfaces respectively for the 12 and 24 elements through the thickness plate. In the model with 24 sub-laminates, a discretization of one ply per element has been adopted. On the other hand, a discretization of two plies per element has been adopted in the model with 12 sub-laminates.

Fig. 8. Model of final test case with each part (left side) and a generic frame of the simulation (right side)

Figures 9 and Figure 10 show the results obtained in terms of contact force, impactor displacement, and impactor velocity as a function of the time for respectively the 12 and 24 elements through the thickness. In the same figures, also the experimental results are reported. It is possible to appreciate the good agreement of the results.

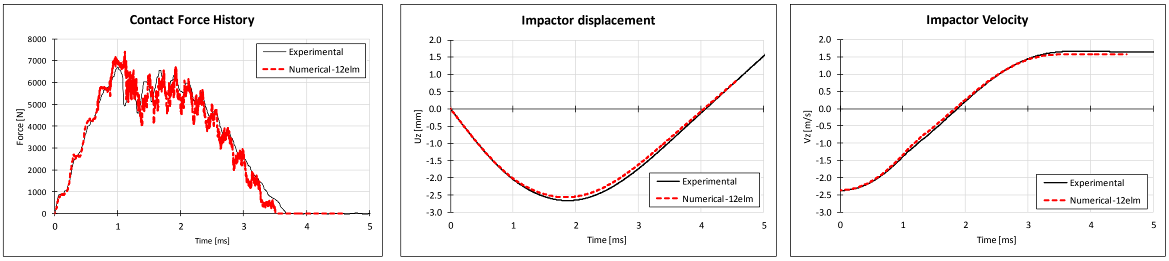

Fig. 8. Model with 12 elements in thickness: contact force (left); impactor displacement (center) and impactor velocity (right)

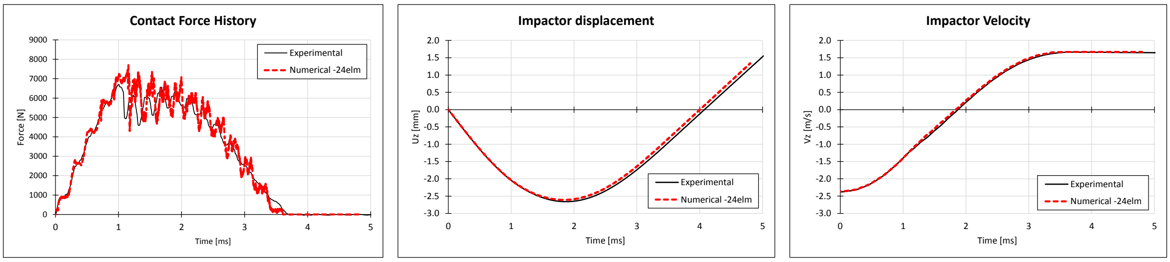

Fig. 9. Model with 24 elements in thickness: contact force (left); impactor displacement (center) and impactor velocity (right)

In Figure 10, a comparison in terms of damaged area is reported. The experimental value, obtained by means of an ultrasonic inspection, is equal to *250 mm*^{2}, while the numerical value is equal to *290 mm*^{2}. In the same image, the damaged elements for tensile fiber and tensile and compressive matrix failure are also reported.

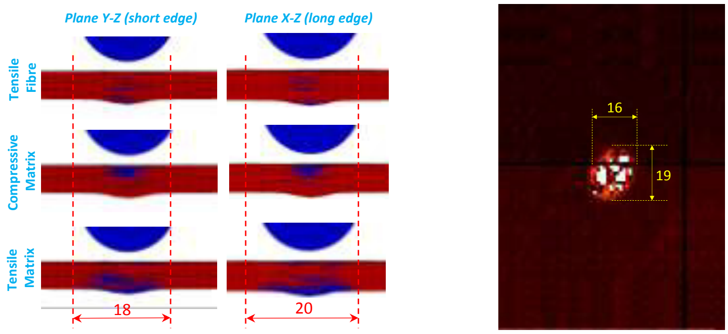

Fig. 10. Model with 24 elements in thickness. Damaged area: numerical (left side) and experimental (right side)

5. Conclusion

This work presents a numerical methodology useful to obtain a good set of numerical parameters for numerical models able to simulate a low velocity impact event on composite structures. In particular, by means of an analytical-numerical correlation, the main characteristics of the numerical model, in terms of elements formulation and element size have been defined. The results show that it is mandatory to use an adequate number of element through the thickness to simulate in the right way the bending stiffness of the plate. The high number of elements through the thickness is finalized also to the simulation of the delamination onset and its evolution. Regarding this aspect, a numerical-experimental procedure has been defined to obtain the most suitable setting for the cohesive zone parameters. The final experimental-numerical correlation demonstrates the reliability of the defined methodologies. The numerical results are very close to the experimental ones, both in terms of contact force and impactor displacement and damaged area.

References

[1] S. Abrate. Impact on composite structures. Cambridge (UK): Cambridge University Press 1998.
[2] C. Li, N. Hu, Y. Yin, H. Sekine, H. Fukunaga. Low-velocity impact-induced damage of continuous fiber-reinforced composite laminates. Part I. An FEM numerical model. Composites Part A 2002; 33: 1055-62.
[3] A. Riccio, S. Saputo, A. Sellitto, A. Raimondo, R. Ricchiuto. Numerical Investigation of a Stiffened Panel Subjected to Low Velocity Impacts. Key Engineering Materials 2016; 665: 277-280.
[4] R. Borrelli, S. Franchitti, F. Di Caprio, F. Romano, U. Mercurio. A numerical procedure for the virtual compression after impact analysis. Advanced Composites Letters 2015; 24(4): 57-67.
[5] A. Riccio, M. Damiano, A. Raimondo, G. Di Felice, A. Sellitto. A fast numerical procedure for the simulation of inter-laminar damage growth in stiffened composite panels. Composite Structures 2016; 145: 203-216.
[6] R. Borrelli, A. Riccio, A. Sellitto, F. Caputo, T. Ludwig. On the use of global-local kinematic coupling approaches for delamination growth simulation in stiffened composite panels. Composites Science and Technology 2015; 115: 43-51.
[7] A. Raimondo, A. Riccio. Inter-laminar and intra-laminar damage evolution in composite panels with skin-stringer debonding under compression. Composites Part B 2016; 94: 139-151.
[8] A. Riccio, S. Saputo, A. Sellitto. A User Defined Material Model for the Simulation of Impact Induced Damage in Composite. Key Engineering Materials 2016; 713: 14-17.
[9] M.R. Wisnom. Modelling discrete failures in composites with interface elements. Compos A Appl Sci Manuf 2010; 41(7): 795-805.
[10] Z. Hashin. Failure criteria for unidirectional fiber composites. Journal of Applied Mechanics 1980; 47: 329-334.