ICILSM 2016
22/26 May 2016
Turin, Italy

Parametric simulation of LVI test onto CFRP plates

A. De Luca[1], R. Sepe[1], G. Lamanna[1], V. Lopresto[2], F. Caputo[1]

[1] *Department of Industrial and Information Engineering, Second University of Naples, Via Roma, 29 – 81031 Aversa, Italy.*
[2] *Department of Chemical, Materials and Production Engineering University of Naples Federico II P.le V. Tecchio, 80 - 80125 Naples – Italy.*

Abstract

The paper deals with the study of the structural behaviour of laminated composite plates under low velocity impacts. Three test cases, respectively with 6J, 10J and 13J impact energies have been experimentally carried out under ASTM D7136 (American Standard Test Method for Measuring the Damage Resistance of a Fiber –Reinforced Polymer Matrix Composite to a Drop-Weight Impact) requirements. Within this work, virtual simulations of such impact tests have been developed by using the finite element code Abaqus®. The numerical model, based on explicit finite element theory, allows predicting the onset and evolution of both inter-laminar and intra-laminar damages. The former have been considered by using special-purpose elements (cohesive elements); the latter thanks to Hashin criteria. For validation purpose, numerical results have been compared with the experimental ones. After the validation phase, a parametric analysis has been numerically performed; the size of the panel support fixture has been considered as main parameter.

Keywords: Low Velocity Impact, Finite Element Analysis, Impact test, Damage

Introduction

Low velocity impacts are damaging phenomena, which frequently affect aeronautical structures. Typical sources of impact are tools falling during manufacturing or maintenance operations, hail, debris on the track, bird collision, etc. Such phenomena heavily influence the mechanical properties of composite material due to the several internal damages, which occur during the impact [1-3]. In particular, the difficulty to survey the damages caused from these events and the difficulty to forecast their failure modes cause the applying of some conservative safety factors to the ultimate load during the design phase. Such safety factors with those related to notch sensibility, environmental conditions and troubles of connection between parts, lower the ultimate load of about 30%. Further ordinary safety factors are applied during the design phase. As a result of this, the ultimate load design value of composite structures is reduced by about 65-70%. This produces a significant increasing in weight and in nominal dimensions of composite structures.
The aim of this work is studying and better understanding the growing mechanisms of low velocity impact damages in order to try to reduce the correlated design safety factor under a damage tolerant design philosophy. At this purpose, some experimental tests have been carried out and numerically simulated.

1. EXPERIMENTAL TESTS

Within this work, some low velocity impacts have been experimentally carried out under ASTM D7136 (American Standard Test Method for Measuring the Damage Resistance of a Fiber – Reinforced Polymer Matrix Composite to a Drop-Weight Impact) requirements. Three series of experimental tests, by considering impact energy values of 6 J, 10 J and 13 J have been performed respectively. In particular, these tests have been carried out onto fiber-reinforced composite specimens with in plane dimensions 100 x 150 mm, with a composite laminate thickness of 2,5 mm, where each

ply is 0.312 mm thick, and lamination sequence is [45/-45/0/90)]s. The material properties are shown in Table 1.
Such experimental tests have been numerically simulated by means of a numerical model based on finite elements theory. To validate the proposed numerical model, numerical results have been compared with data from experimental tests. Once the validity of the model has been checked, a parametric analysis has been numerically performed by changing the support fixture and in particular, the size of the panel unsupported zone.

Table 1: Material proprieties.

Longitudinal Young's Modulus E_{11}	130.05	GPa	Longitudinal Tensile Strength X_t	1460.7	MPa
Transverse Young's Modulus E_{22}	11.55	GPa	Longitudinal Compressive Strength Xc	876.42	MPa
In Plane Shear Modulus G_{12}=G_{13}	6	GPa	Transverse Tensile Strength Yt=Zt	77.145	MPa
In Plane Shear Modulus G_{23}	6	GPa	Transverse Compressive Strength Yc=Zc	241.44	MPa
Poisson's ratio $\nu_{12}=\nu_{13}$	0.312		Shear Strength S_{12}=S_{13}	90	MPa
Poisson's ratio ν_{23}	0.48		Shear Strength S_{23}	40	MPa
Critical ERR-MODE G_{IC}	180	Jm^{-2}	Critical ERR-MODE II-III $G_{II-IIIc}$	500	Jm^{-2}

2. FE MODEL

The numerical model has been realized by using Abaqus/Explicit® software where, for reducing the required CPU time, a Global-Local approach has been used to model the composite plate specimen. Local zone allows predicting the onset and evolution of both inter-laminar and intra-laminar damages, while global zone only the intra-laminar ones. Inter-laminar damages have been considered by using special-purpose elements (cohesive elements). In particular, in the impact zone each lamina has been modeled with one layer of plane finite elements in thickness and, between each pair of laminate, cohesive elements have been placed. Intra-laminar damages have been considered by applying a Progressive Failure Analysis (PFA) techniques. Among various PFA algorithms implemented in Abaqus/Explicit® software, Hashin criteria have been chosen. Such criteria are based on the separation of different failure modes and allow predicting the onset and the evolution of the fiber and matrix failures. The only one difference between global and local zones is that in the global zone there are not cohesive elements layers. In more detail, continuum shell elements (SC8R from the Abaqus elements' library) have been used to model the composite plies, cohesive elements (COH3D8 elements from the Abaqus elements' library) to model the inter-lamina interface and solid elements (C3D8R elements from the Abaqus elements' library) to model the drop mass and the support fixture. The choice to use continuum shell elements to model the composite laminate has allowed defining the composite stacking sequence by means of "Composite Layout Abaqus®" tool. As matter of the fact, 190228 elements (166786 nodes) have been used in the model, whose 121600 are continuum shell elements, 19986 are Solid elements and 25200 are cohesive elements. Moreover, the continuity between the Local zone and Global zone has been ensured by linking them via "tie constraints" elements. This technique allows linking all degrees of freedom of a geometrical surface to another surface. In this case, the node to surface contact is used in order to improve numerical convergence of results. In Figure 1.a the FE model is shown; the plate has been constrained both in plane and out of plane, the drop mass is able to move only along out of plane direction and the support fixture has been completely constrained. Figure 1.b shows the parameterized support fixture and the unsupported panel zone.
In the Figures from 2.a to 2.d the numerical results for the test case at 10J are compared to the experimental ones. In particular the drop mass-coupon interface contact force (Figures 2.a), the impact energy (Figure 2.b), the drop mass velocity (Figure 2.c) and the drop mass displacement (Figure 2.d) are plotted as function of the time and for different boundary conditions (in the case of results plotted in figures 2, fixture extension has been changed).
According to Figure 2, experimental curves are in a good agreement with the respective numerical ones. Such good agreement allowed using the numerical model in order to perform a parametric

analysis only by using finite element techniques. The unsupported zone has been numerically changed as aforementioned. From such analysis, it has been possible to show how the structural behavior of the composite panel changes by considering different support fixtures. As expected, by reducing the area of the unsupported zone the panel behavior appears stiffer.

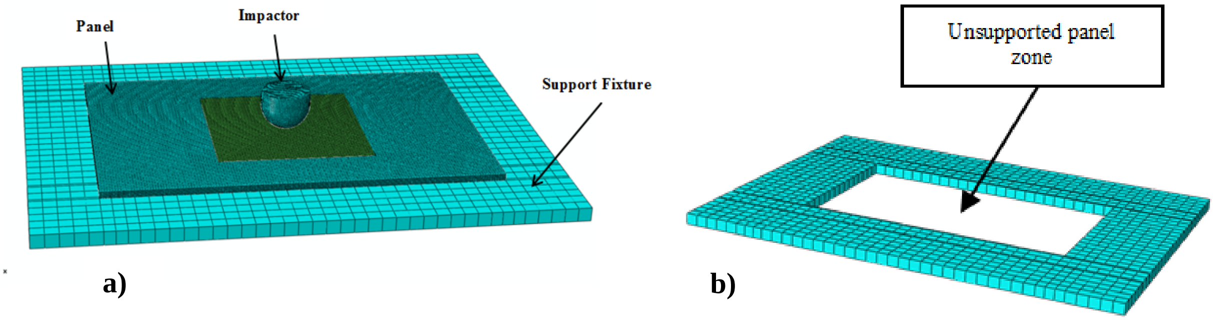

Figure 1 – a) FE model; b) support fixture.

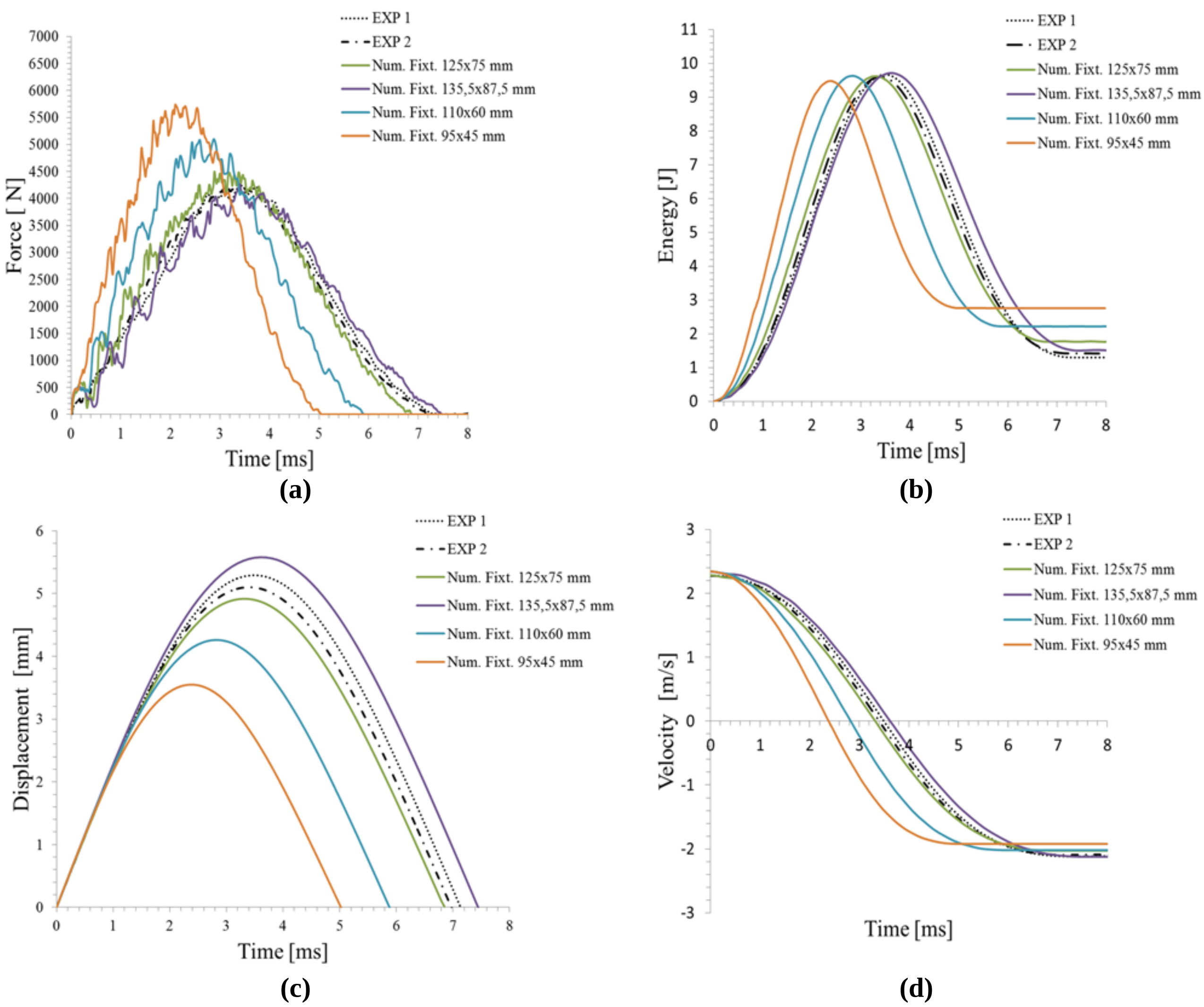

Figure 2 – Numerical and experimental results: a) force vs. time; b) energy vs. time; c) displacement vs. time; d) velocity vs. time.

In Tables 2, for the case at 10 J, the comparison between the numerical and experimental data, for the various support fixtures, in terms of peak value of the impact contact force, is shown.
Results from the cases at 6 J and 13 J are in good agreement with the experimental ones too.

Table 2: Comparison between numerical and experimental data for the 10 J impact test.

LVI 10 J	Unsupp. Zone size [mm]	F_{max} [N]	$\Delta F_{max\ I}$ [%]	$\Delta F_{max\ II}$ [%]
Num. Standard fixture	125x75	4498.19	6.28	5.50
Num. Support Fixture 1	133.5x87.5	4266.86	8.14	0.07
Num. Support Fixture 2	110x60	5099.12	20.47	19.59
Num. Support Fixture 3	95x45	5735.87	35.52	34.53
Experimental test I	125x75	4232.42		
Experimental test II	125x75	4263.77		

The delamination damage has been well numerically predicted both in shape and in size. In particular, the correlation of numerical and experimental data has been shown in Table 3.

Table 3: Comparison between numerical and experimental data for the 10 J impact test.

Impact test 10J	Numerical values [mm]	Average experimental values [mm]	Percentage difference [%]
Major axis	17	17.13	0.76
Minor axis	13	12.39	4.92

Figure 3 shows how the delamination area changes in shape and size by modifying the support fixture on which the panel is arranged: reducing the panel unsupported zone, delamination area increase.

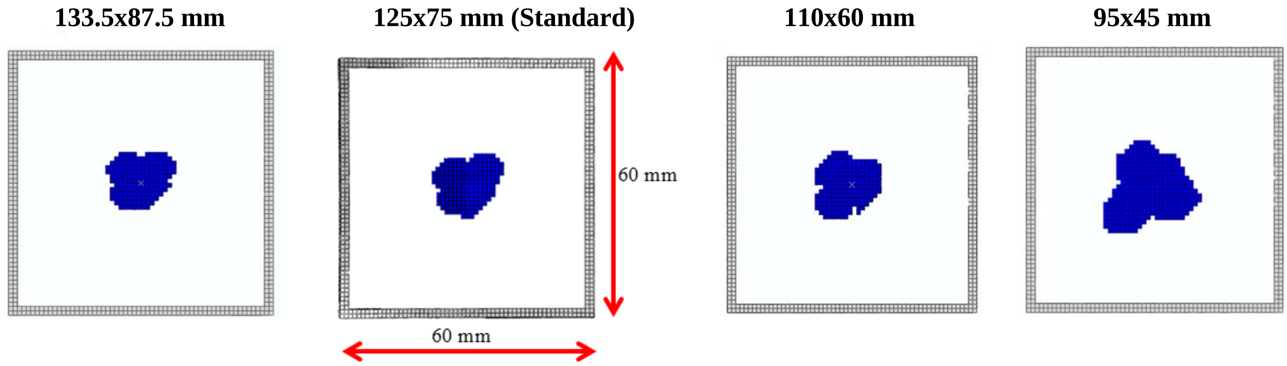

Figure 3 – Delamination area by varying the support fixture.

3. Conclusions

The validity of the presented model has been confirmed by the application of the same one to experimental tests with impact energy equal to 6 J, 10 J and 13 J and the Global-Local numerical modeling approach worked well and it allowed saving considerable CPU time without influencing the quality of results. These numerical studies have provided very interesting results regarding Low Velocity Impact damaging mechanism of a composite plate under different boundary conditions.

References

[1] Caputo C., De Luca A., Sepe R., *Numerical study of the structural behaviour of impacted composite laminates subjected to compression load*, Composites Part B, **Vol. 79**, pp. 456-465, 2015.

[2] Caputo F., De Luca A., Lamanna G., Lopresto V., Riccio A., *Numerical investigation of onset and evolution of LVI damages in Carbon-Epoxy plates*, Composites Part B, **Vol. 68**, pp. 385–391 2015.

[3] Riccio A., De Luca A., Di Felice G., Caputo F., *Modelling the simulation of impact induced damage onset and evolution in composites*, Composites Part B, **Vol. 66**, pp. 340–347, 2014.