A GLOBAL-LOCAL APPROACH TO DESCRIBE THE DAMAGE PROPAGATION IN COMPOSITE LAMINATED STRUCTURES

A.Riccio[1], F. Caputo[1*], E. Antonucci[2], V. Lopresto[3], M. Zarrelli[2]

[1]Dipartimento di Ingegneria Aerospaziale e Meccanica – Seconda Università degli Studi Napoli – via Roma, 29 – Aversa (CE) – Italy –
[2]CNR-Research National Council, IMCB – Inst. of Comp. and Biom. Mat.- P.E. Fermi – Portici (NA) – Italy -
[3]Department of Materials and Production Engineering - University of Naples "Federico II" – P.le Tecchio, 80- Naples, Italy –
**francesco.caputo@unina2.it*

Keywords: Composite · Low Velocity Impact · Finite Element Analysis · Damage

Abstract

In the last years meaningful improvements have been made in understanding failure mechanisms of composite materials.Delamination and other damage mechanisms, such as matrix cracks, fibre-matrix debonding and fiber failure, can appear as a consequence of impact events with foreign objects, under service conditions and maintenance operations. These phenomena are seldom analyzed together without discussing how the interferences between the different damage mechanisms can influence their evolution under different loading conditions.

In the present work, the activities will be focused on the development of a specific numerical procedure, able to describe the failure modes of composite structures subject to a low velocity impact. A very fine mesh refinement is required in the impacted area, where the impact induced damage will onset. In order to reduce the computational cost without compromises on the accuracy of results, a global/local approach has been adopted: a very refined mesh has been used in the critical region, whereas a coarser mesh has been used in the rest of the domain. In the present work Multi-Point-Constraints (MPS) has been used to link the local domain to the global domain without using transition meshes. The implementation and the analyses have been performed in the ABAQUS® FE code.

1.1 Introduction

The impact damage influences the mechanical properties of composite materials generally constituted by components (such as fiber and matrix) whose stiffness and strength-to-failure can be extremely different. Composites are characterized by several interacting failure modes such as matrix breakage, fiber failure and delaminations. These failure modes, which can be simultaneously induced by low velocity impacts, can be very difficult to detect by visual inspections on the structure. Hence non-destructive techniques, such as ultrasound based techniques must be adopted for the inspection of composite materials even if they are generally slow, as well as expensive, and technical difficulties can arise during their application.

The aeronautical structures are designed assuming preexisting impact damages under BVID (Barely Visible Impact Damages) threshold, determining a drastic reduction of composites structure's resistance, which leads to a significant increase in weight. The allowables of damaged structure are determined through long and expensive experimental tests. In order to reduce the costs associated to the needed experimental campaigns it's very important to integrate tests with numerical tools, able to help understanding the structural behavior under complex loading conditions.

The aim of this work is to develop a modeling technique capable to predict the damage onset and evolution in composites due to low velocity impacts. The modeling technique is able to simulate the initiation and propagation of inter-laminar and intra-laminar damage.

2.2 Composite failure modeling

Inter-laminar and intra-laminar failure mechanisms are taken into account in this paper. Intra-laminar damages such as the breaking of the fibers, the matrix cracking and the interface damages between fibers and matrix (debonding) are generally caused by in-plane layer stresses and take place in each single layer. On the other hand, the inter-laminar damage such as the delamination between different layers is generally caused by out-of-the-plane stress components. Intra-laminar and inter-laminar damage mechanisms can onset and evolve independently, or may interact each other.

The introduced numerical model has been implemented in the Abaqus Explicit Solver®. The intra-laminar damage onset and evolution is taken into account by stress based failure criteria and material degradation rules. The delamination onset and propagation is modeled by means of cohesive elements. These two different approaches are briefly described hereafter.

The progressive damage method, for intra-laminar damage onset and propagation, implemented in the Abaqus code, offers a general capability of modeling progressive damage and failure in fiber-reinforced composites in terms of fiber and matrix failures. Four different modes of failure are considered: fiber rupture in tension and compression, and matrix cracking under transverse tension and compression. The constitutive relation adopted for each failure mode is schematically shown in figure 1.

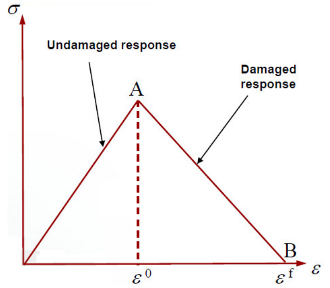


Figure 1: constitutive relation adopted for each failure mode

The undamaged constitutive behavior is defined as orthotropic elastic. An initialization phase and a propagation phase can be distinguished. When the element stress exceeds a limit value, the element is considered partially damaged and the damage propagation starts. The point A in figure 1 shows the damage initiation of stiffness degradation. The damage initiation criteria for fiber reinforced composites are based on Hashin's theory [1-2].

$\left(\frac{\sigma_{11}}{X_t}\right)^2 + \alpha\left(\frac{\tau_{12}}{S_l}\right)^2 = 1$	Fiber Tensile failure
$\left(\frac{\sigma_{11}}{X_c}\right)^2 = 1$	Fiber Compressive failure
$\left(\frac{\sigma_{22}}{Y_t}\right)^2 + \left(\frac{\tau_{12}}{S_l}\right)^2 = 1$	Matrix Tensile failure
$\left(\frac{\sigma_{22}}{2S_t}\right)^2 + \left[\left(\frac{Y_c}{2S_t}\right)^2 - 1\right]\left(\frac{\sigma_{22}}{Y_c}\right) + \left(\frac{\tau_{12}}{S_l}\right)^2 = 1$	Matrix Compressive failure

Table 1 Hashin’s Failure criteria

The path A-B shows the damage evolution that defines the post damage-initiation material behavior. It describes the rate of degradation of the material stiffness once the initiation failure criterion is satisfied. For the damage propagation, a linear evolution law has been considered. The law is based on the energy dissipated during the process. An element is removed (deleted) once the damage variables for all failure modes at all material points in the element reach the maximum degradation value (Dmax). A similar damage evolution based on the fracture energy is used in cohesive elements theory for the simulation of delamination growth.

The constitutive modeling of cohesive elements is based on a traction-separation description for delamination. Linear elasticity with damage and non-standard constitutive laws are used. The constitutive response in cohesive elements for delamination applications is characterized by an initial damage phase, a damage evolution phase and the possibility to remove full damaged elements. The cohesive damage evolution phase, representing the post-damage initiation response, is based on two criteria (energy criterion and displacement criterion). Hence, the total fracture energy must be specified when the energy criterion is used, while the post-damage ultimate displacement at failure of the element must be defined for displacement criterion. The fracture energy is defined as the area under the constitutive response curve. The failure displacement is the last possible displacement before the cohesive element is totally damaged. In this work the damage evolution based on the fracture energy criterion has been adopted.

3 FE modeling and analysis

The focus of this paper, is to use the implemented progressive damage approach to simulate the damage onset and propagation in a stiffened panel subjected by a low velocity impact. The analyzed stiffness panel is a typical aeronautical structure and the impact is simulated with a rigid impactor of fixed kinetic energy

However, a pilot analysis on a very simple test case has been carried out, first, for a preliminary validation of the implemented progressive damage procedure against literature experimental results [1]. The simulations have been carried out by Abaqus Explicit Solver®. The full-scale panel FE model has been built by means of the Pre/Post Abaqus CAE®.

3.1 Preliminary test case description and validation of the Progressive Damage Model

The reference experimental test to preliminary validate the implemented progressive damage procedure is reported in [1]. The test-case consist in a low velocity impact on a clamped plain panel. The impactor is modeled as an analytical rigid surface with a body diameter of 16 mm, a mass of 2.63 Kg and a constant impact velocity of 3.9 m/s. The mass and velocity are chosen in order to obtain the prescribed impact energy. The total thickness of plate is 3mm . The material system adopted to manufacture the plate is IM7/977-2. The lamina material properties for this material system are shown in table 2.

Longitudinal tensile modulus	165.1 GPa	Longitudinal tensile strenght	2594.0 MPa
Longitudinal compression modulus	135.2 GPa	Longitudinal compressive strenght	1120.9 MPa
Transverse tensile modulus	8.3 Gpa	Transverse tensile strenght	81.0 MPa
Transverse compression modulus	8.4 GPa	Transverse compressive strenght	223.2 MPa
Poisson ratio 12	0.27	Interlaminar shear strenght	114.0 MPa
Poisson ratio 13	0.25	Poisson ratio 23	0.41

Table 2 Material properties

In order to reduce the computational time needed for the analyses without losing accuracy of results, a Global-Local approach is used to create the Finite Element model of the composite plain panel. A refined local finite element model of the impact area has been linked to a coarser global model of the rest of the panel by adopting the ABAQUS "tie constraints" option based on multi-point-constraints. This technique allows matching all degrees of freedom of a surface to the ones of another surface. In this case the "node to surface" contact algorithms is used. The refined FE model is characterized by one shell element per ply with cohesive elements placed at each interface between two consecutive plies to simulate the bonding layer. Contact elements are used between shell continuum elements and cohesive elements, to fix connection between separated layers. The plate is composed by 24 plies, $[(-45,+45,90,0)_3]_s$, and 23 layers of cohesive elements between each couple of plies, to be able to model inter-laminar damage onset and evolution. The thickness of each cohesive layer is set to 0.001 mm in order to avoid significant changes in the total thickness of the plate. The Shell Continuum (SC8R) and Cohesive (COH3D8) elements have been used. Removal element capability has been enabled. The boundary conditions applied to the panel simulate the impact machine gigs. Clamping conditions (UX=UY=UZ=0) have been imposed to the panel nodes which are placed outside the 100 mm diameter ring fixture. Between impactor and plate surface to surface contact elements are used. Figure 2 shows the FE boundary conditions and the used impactor properties.

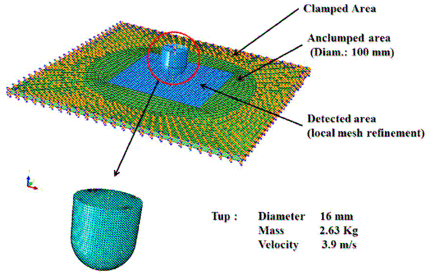
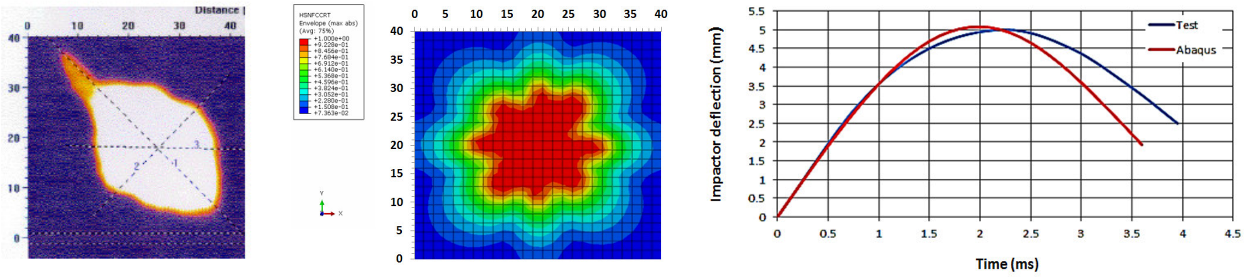


Figure 2: Boundary condition and impactor properties

In [1] the introduced experimental damaged area scans [1] do not indicate the different types of failure (delaminations, matrix cracks, fiber fracture) at the different plies; hence the numerical-experimental correlation is made in figure 3 on the damaged area as an envelope.
The numerical results figure 3.b show the elements which have failed at least in one ply for at least one of the failure criteria. The contour plot takes into account the value of the criterion (criterion =1 for plies completely broken and criterion <1 for ply not completely broken).
The numerical results are compared with the experimental C-scan (figure 3a) obtained after the low velocity impact test showing a very good agreement in terms of enveloped damaged area (exp: 481.2 mm^2 Vs. num: 470.6 mm^2).
In figure 3.c the numerical-experimental comparison in terms of impactor deflection vs. time curves is shown. Again a very good agreement between numerical results and experimental data is obtained.



(a) (b) (c)

Figure 3: (a) Experimental damage area; (b) Numerical damage area; (c) time-tup deflection

The comparisons above shown demonstrate that the implemented progressive damage model is very effective and accurate in predicting both the damaged area and the dynamics of the low velocity impact test.

In figure 4 the delaminations distribution along the thickness is shown. The different delaminations, with peanuts shape, arising at the interfaces between plies as a consequence of the low velocity impact event are clearly visible.

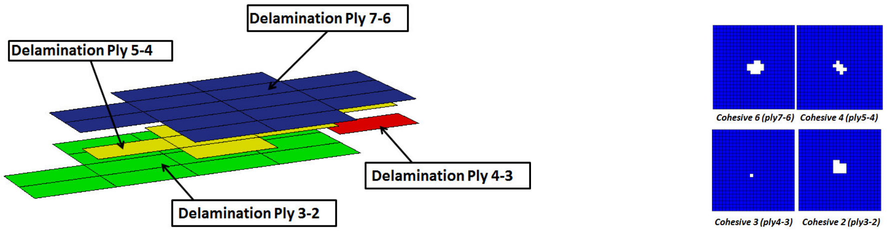


Figure 4 delaminations distribution along the thickness

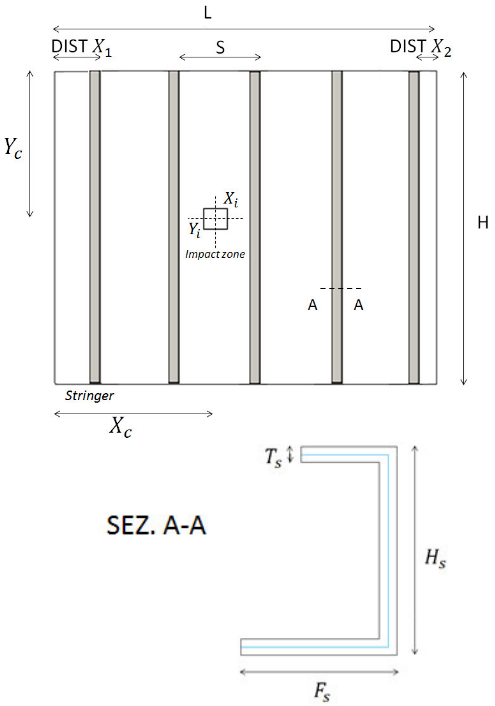
In figure 5 the intra-laminar damage distribution along the thickness is shown. The fibre and matrix failure in each ply as a consequence of the low velocity impact event can be appreciated.

(a) (b)

Figure 5: (a) Fiber tensile failure; (b) Matrix tensile failure

3.2 Stiffened panel test-case

Figure 6 schematically shows the geometrical description of the stiffened test-case panel, subjected to a low velocity impact, considered in the frame of this study.



Parameter	Value [mm]	Description
L	946.16	Panel length
H	900	Panel height
X_c	450	X coordinate impact zone
Y_c	360.56	Y coordinate impact zone
S	215	Feet stringer distance
Dist X_1	56.56	Distance stringer-edge panel
Dist X_2	29.59	Distance stringer-edge panel
X_i	60	Impact zone length
Y_i	60	Impact zone height

Parameter	Value [mm]	Description
F_s	24	Wight of stringer cap
H_s	33	Stringer height
T_s	2	Stringer thickness
C_s	18.5	Wight of stringer foot

Figure 6: Stiffened panel subjected to a low velocity impact – Geometrical description

The impactor is again modeled as an analytical rigid surface with a body diameter of 16 mm, mass of 5.26 Kg and a constant impact velocity of 3.9 m/s (impact energy 40J).
The material system considered for this stiffened panel is again of plate is IM7/977-2. The test panel is 900x946.16 mm wide with a skin thickness of 2 mm (being each ply 0.167 mm thick). A Global-Local approach is used to model the stiffened composite panel in a similar as the preliminary test-case. A refined model in the impact zone and a coarser one for the rest of the panel have been introduced. In figure 7 a view of the FE model is presented highlighting the global local approach used to model the impact area (60x60 mm).

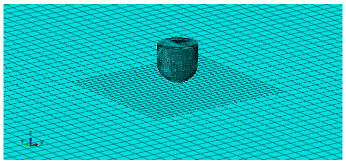


Figure 7: Global-Local approach

The panel skin is composed by 12 plies with staking sequence $[90,0,+45,90,-45,0]_s$ and 11 layers of cohesive elements between each couple of plies to model inter-laminar damage. The thickness of cohesive elements is again 0.001 mm. The finite elements used to model the panel are again the Continuum (SC8R) and Cohesive (COH3D8) elements. The stringers are modeled with conventional shell elements with associated the same stacking sequence as the skin.A Dynamic Explicit structural analysis has been performed for the simulation of the impact event on the stiffened panel and both intra-laminar and inter-laminar failure mechanisms have been considered. FE Constraints are clamping conditions (UX=UY=UZ=0) at stringer feet. From the numerical simulation it is possible to trace the areas where each ply fails respect to Hashin criteria (Intra-laminar failure). In figure 8 and figure 9, the Abaqus Explicit Output failure indexes for fiber compressive failure obtained for the Ply 12 and 11 (ply 12 is the first ply on the impactor side) are, respectively, shown. The step time is the last impact step (just before the end of the contact phenomenon between the impactor and the plate).

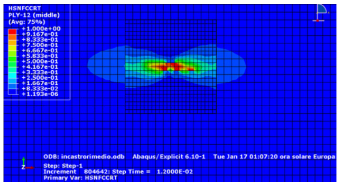


Figure 8: Ply 12 - fiber compressive failure

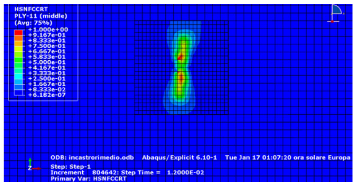


Figure 9: Ply 11 - fiber compressive failure

In figure 10 the delaminations distribution along the thickness is shown in the impacted area of the stiffened panel. The different delaminations, with peanuts shape, arising at the interfaces between plies as a consequence of the low velocity impact event are clearly visible. In figure 11 the delamination arising between plies in the impacted area are shown in a 3D view.

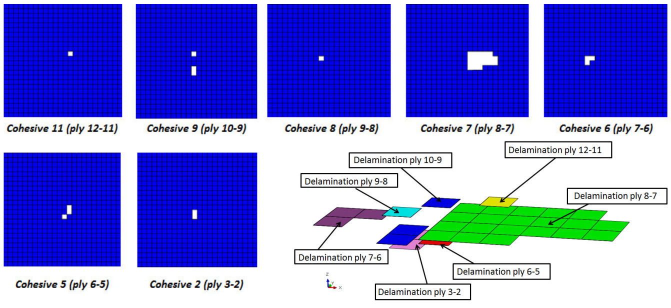


Figure 11: Cohesive element failure

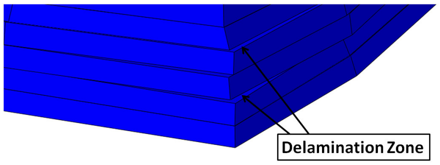
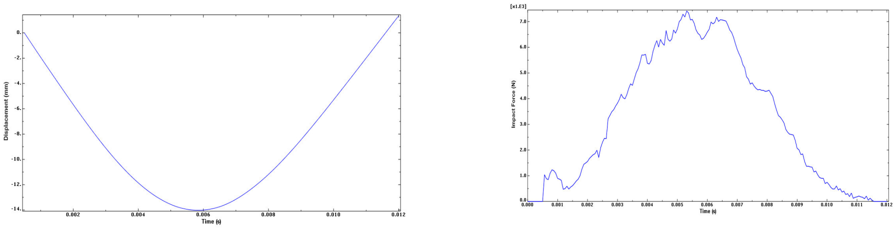


Figure 12: Delamination zone between the plies

Figure 13 shows the displacement (force) of the impactor as a function of the time.



(a) (b)

Figure 13 : (a) Displacement; (b) impact force of the impactor as a function of the time

The following conclusions can be extrapolated from the above Low Velocity Impact Simulation on the stiffened panel:

- The prediction of the damaged area is detailed both in terms of intra-laminar and inter-laminar damge propagation.
- the considered FEM analysis methodology allows to evaluate the impact damage accurately even in the presence of very complex structure by means of a very effective global local approach.

4 Conclusions

In this paper a very effective progressive damage model in introduced. The model is able to estimate with a good approximation the damage onset an propagation due to an impact event on complex composite structures. The prediction of the impact response and the damage onset and propagation in laminates were performed by using the Abaqus Explicit Solver®. The implemented failure criteria allow to predict delamination growth (cohesive element criterion) and intra-laminar damage onset and propagation. A Global-Local Approach is used to effectively link separated models built for the impact area (refined model) and for the rest of the panel (coarse model), obtaining advantages in terms of CPU spent without losing results accuracy. A preliminary validation has been carried on a simplified test case. Numerical results have been compared to literature experimental data obtaining a very good agreement in terms of impact induced damage area.

The implemented progressive damage approach has been applied to a stiffened composite panels subjected to a low velocity impact. Results in terms of damage propagation and impact forces vs. time curves show that the approach is able to accurately predict the damage propagation and could be effectively used to integrate experimental activity on low velocity impacts. Finally the performed work has contributed to improve the understanding of the damage phenomena in impact zones in complex composite structures.

REFERENCE

[1] GARTEUR AG-28: IMPACT DAMAGE AND REPAIR OF COMPOSITE STRUCTURES -GARTEUR / TP – 155 September 2006

[2] Hashin Z. Failure criteria for unidirectional fiber composites. Journal of Applied Mechanics 1980; 47: 329-334

[3] Abaqus 6.10 Documentation

[4] Abaqus 6.10 Theory Manual

[5] Liu D., Impact induced delamination - a view of bending stiffness mismatching. Journal of Composite Materials, 22(7), pp.674-692, 1988

[6] Modellazione della resistenza post-impatto di laminati in composito, Cattaneo, Politecnico di Milano, Maggio 2010

[7] Pagano, N. J., “Exact Solutions for Composite Laminates in Cylindrical Bending,” Journal of Composite Materials, vol. 3, pp. 398–411, 1969.

[8] Karan, S. S., and R. M. Sorem, “Curved Shell Elements Based on Hierarchical - Approximation in the Thickness Direction for Linear Static Analysis of Laminated Composites,” *International Journal for Numerical Methods in Engineering*, vol. 29, pp. 1391–1420, 1990.

[9] Lekhnitskii, S. G., Anisotropic Plates, translated from second Russian edition by S. W. Tsai and T. Cheron, Gordon and Breach, New York, 1968

[10] Spilker, R. L., S. Verbiese, O. Orringer, S. E. French, E. A. Witmer, and A. Harris, “Use of the Hybrid-Stress Finite-Element Model for the Static and Dynamic Analysis of Multilayer Composite Plates and Shells,” *Report for the Army Materials and Mechanics Research Center*, Watertown, MA, 1976.

[11] Ferencz, R. M., “Element-by-Element Preconditioning Techniques for Large-Scale, Vectorized Finite Element Analysis in Nonlinear Solid and Structural Mechanics,” Ph. D. Dissertation, *Stanford University, Stanford*, CA, 1989.

[12] Johnson, G. R., and W. H. Cook, “Fracture Characteristics of Three Metals Subjected to Various Strains, Strain rates, Temperatures and Pressures,” *Engineering Fracture Mechanics*, vol. 21, no. 1, pp. 31–48, 1985.

[13] Backman, M. E., and W. Goldsmith, “The Mechanics of Penetration of Projectiles into Targets,” *International Journal of Engineering Science*, vol. 16, pp. 1–91, 1978.

[14] DELAMINATION GROWTH PREDICTION USING A FINITE ELEMENT APPROACH P. ROBINSON, T. BESANT & D. HITCHINGS, *Department of Aeronautics, Imperial College of Science, Technology & Medicine*, London, UK