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.

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

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) to each other. Hence, in this case no contact formulation is needed, reducing the calculation time. In the TIE model each single ply is modelled separately, and then the different laminae are stacked and joined through the tie contact. Nodes in contact are tied with a master-slave algorithm that ensures the rigid bonding of the surfaces, thereby almost halving the computational time. Furthermore, these two models are only able to evaluate the intra-plies failure. They do not consider the delamination phenomenon and if delamination occurs experimentally, they tend to overestimate the ultimate load. Only the COH model takes delamination damage into account. As in the TIE model, the single laminae are modelled separately and are then joined through a contact formulation, which in this case is the cohesive surface based interaction which is used to model the inter-laminar damage. Analysis which take delamination results into account are the slowest. The damage onset and the evolution of delamination are evaluated through the QUADS and B-K criteria respectively. Concerning the intra-plies failure, a Hashin criterion was implemented, however in ABAQUS only a 2D Hashin criterion is available and it works only with shell or continuum shell elements. Hence, a user-subroutine was used in this work to overcome this issue and to ensure the possible use of solid elements.

3.1. Mesh

The mesh is another important aspect of simulations. It is necessary to find a compromise between a coarse mesh to ensure a non-excessive calculation time, but fine enough mesh to correctly and faithfully reproduce the simulated phenomenon. In the present work, different meshes were tested to study their effect on the solution. Particularly three different dimensions were used, indicated as m1, m2 and m3 as reported in Table 2 for each kind of test. The value reported for the open hole are referred to the near hole area whereas away from hole a coarser mesh was used. In general, and unless otherwise specified, one element per lamina in the thickness direction was used. As shown below it was found that mesh size mainly affects only the breaking load but does not influence the stiffness.

Table 2. Mesh size

	Tension	Open hole
m1 [mm]	5	2
m2 [mm]	3	1
m3 [mm]	1	0.5

3.2. Boundary conditions and constrains

In this section, the boundary conditions for the different loading cases are investigated. In the experimental tests, the crosshead movement was applied in displacement control and it was simulated through the definition of a reference point (RP) to which the motion law was assigned. In case of the tensile simulation, the movement was transmitted to the specimen through a coupling interaction between the RP and one side of the specimen, whereas the other side was coupled to a second reference point, whose position was fixed. The reaction force was registered in the static reference point and the displacement results from the difference between the displacement of other two reference points placed in the middle of the specimen at a distance of 25mm, to simulate the measurement of the extensometer. A variety of different ways to simulate the gripping area were tested in order to find a satisfactory solution for all the model configurations (PB, TIE, COH). In reality, the gripping system applies a certain pressure on the specimen and therefore the sliding of each ply is reduced. In the FE model, such pressure was not modelled because experimentally it is hard to acquire its exact value. The consequence is that in case of PB it is sufficient to apply the coupling on the two outer surfaces in the gripping area of the specimen whereas in case of TIE and COH this leads to a premature and wrong failure of the laminates. Practically, without pressure the transmission of the load between plies do not work very well leading to an overload of the two outer surfaces which fail in a premature way. In order to overcome this issue, the gripping area was not modelled but only the effective length of the specimen outside the gripping was modelled and a coupling on the entire transversal area at the two ends of the specimen was imposed.

3.3. Material calibration

The first parameters to be checked are the material properties in order to fit the numerical results with the experimental ones: the numerical models have to be able to correctly reproduce the laminate response in terms of the slope of the load-displacement curves and the failure load. Except for the few data obtained from the UD experimental tests (fibre failure properties and UD elastic modulus), the complete determination of the material behaviour was based on data assumed or deduced from other similar materials present in the literature or from technical datasheets of the manufacturers of fiber and matrix, Cicco (2016), Anonymous (2013), Ridgard (2008). In the present activity, the focus was placed on the assessment of the fracture strength of the material. Indeed, the matrix mode failure strength (in compression and in tension), were tuned because the literature data are not adequate for a very refined simulation and experimental tests concerning the matrix are neither available nor easy to perform. These calibrations were performed on the balanced specimen using solid elements. In order to save computational time and due to the great number of tests required to perform for fine tuning, the simplest modelling technique PB and a 3mm mesh was adopted. However, the consistency of the results with those of more complex models was verified. In Fig.1 the comparison between the experimental, analytical (classic lamination theory CLT, Gibson (2016)) and numerical (PB, mesh 3mm) results for both unidirectional and balanced specimens is reported. In the unidirectional case, there was only a minor mismatch between the experimental and both the analytical and numerical curves showing only a slightly lower slope.

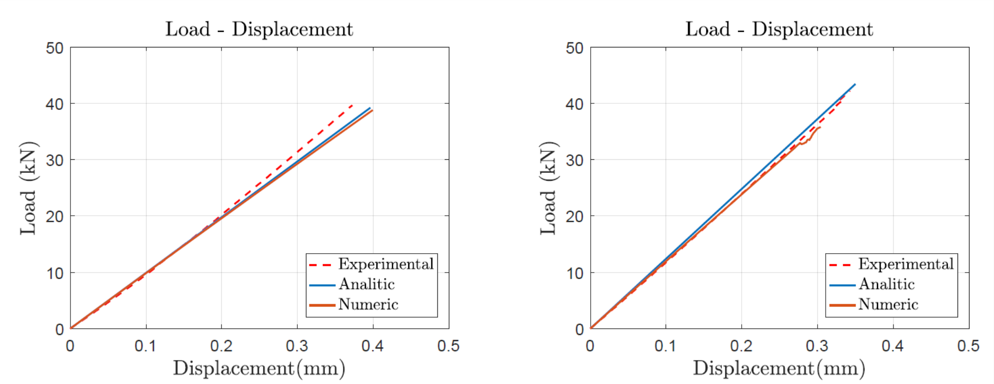


Fig. 1. Uniaxial tensile test. Comparison between the analytical, numerical and experimental load-displacement curves (a) UD specimen (b) balanced specimen

Also in the balanced case, there is a good agreement with experimental data with regards to stiffness, but some discrepancies are visible in the last part of the curve. It is evident from the literature that the epoxy matrix tensile and compression strength have a high variability. Therefore, five different material combinations slightly modifying the nominal strength properties reported by the manufacturers were studied. In Table 3, the material parameter set for the damage investigated are reported, while in Fig.2 the results for the balanced specimen are shown. A variation of the strength values of a few percentage points led to changes in only the last part of the load-displacement curve.

Table 3. Material data

Property	Symbol	Nominal	Mod 1	Mod 2	Mod 3	Mod 4
Fibre strength in tension	Xt [MPa]	2643.33	2643.33	2643.33	2643.33	2643.33
Fibre strength in compression	Xc [MPa]	1405	1405	1405	1405	1405
Matrix strength in tension	Yt [MPa]	63,8	78	73	75	70
Matrix strength in compression	Yc [MPa]	169.38	192.78	192.78	192.78	192.78
Shear strength	S [MPa]	103,8	103,8	105	109,5	114

The fibre strength in tension of the fiber was left unchanged because it was obtained from experimental unidirectional tensile tests and also all the elastic modulus remained unchanged. The tuning was devoted only on the matrix strength in order to find a material set leading to failure at the desired load. The "Mod 1" configuration appeared to be the most promising. In the Mod 1 configuration, the matrix tensile and compressive strength were increased by about 20% and 10% respectively compared with the reference value. In order to ensure that the chosen data are adequate, a further validation on a different component is required. Such a comparison was carried out with the results of a uniaxial tensile test on a balanced specimen having a central hole. The results reported in Fig.3 show that even for the specimen with a hole, the tuned material data leads to very satisfactory behaviour of the FE curve which reproduces the experimental data well.

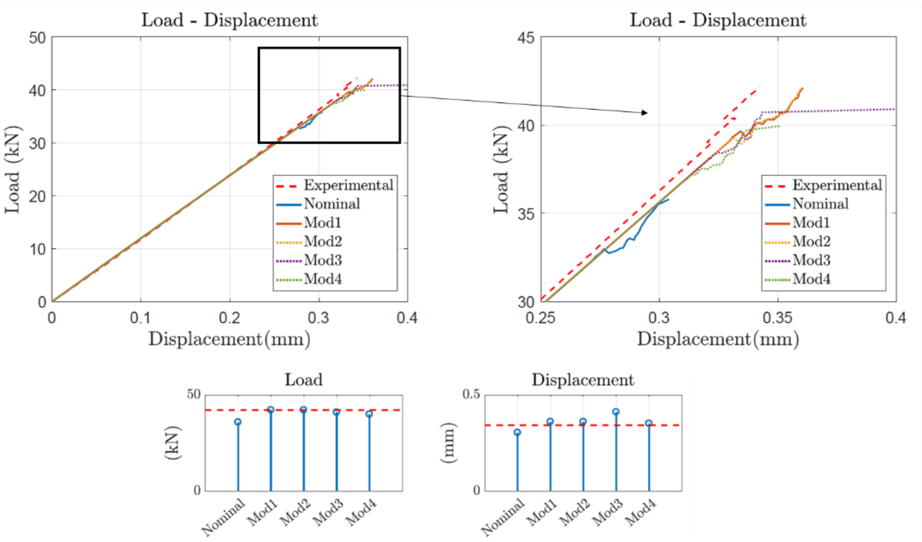


Fig. 2. Load-displacement curves of the balanced specimen changing the failure parameter inside the literature variable range. On the right, a focus on the last part of the curve

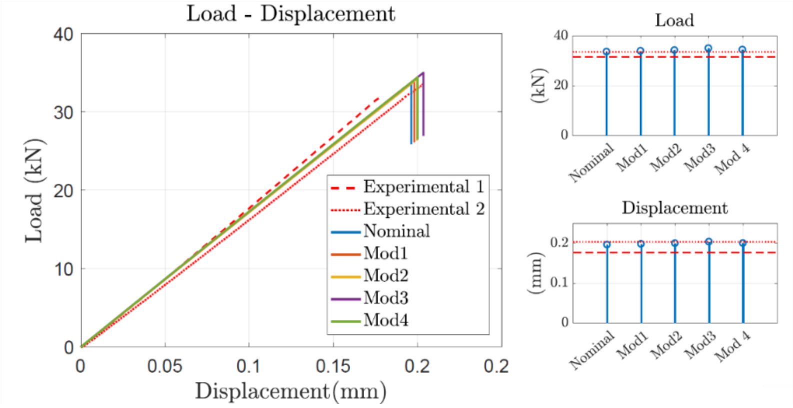


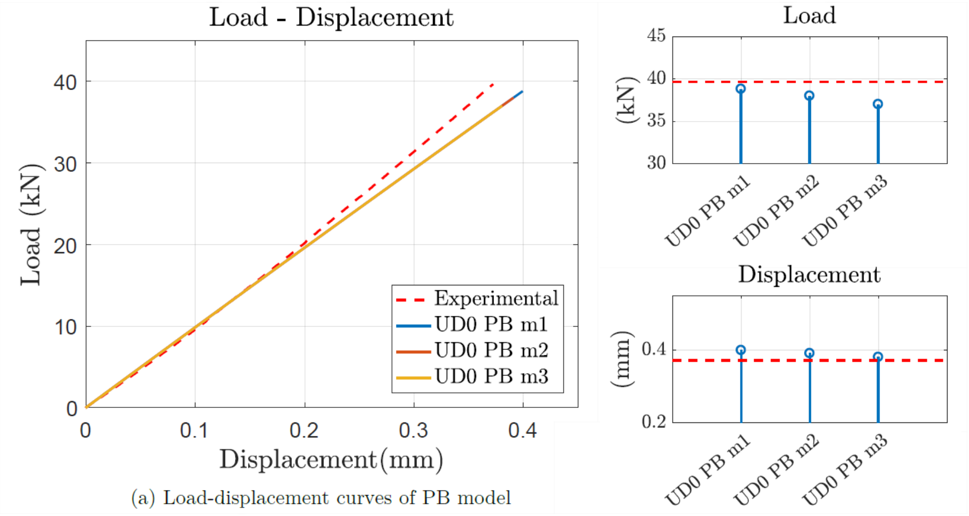
Fig. 3. Comparisons of the load-displacement curve between the FE model and the experiments of a balanced with hole specimen. Different values for failure were investigated

4. Results

In this section, the results of the tensile test simulation for unidirectional and balanced specimens are reported focusing on the effects that both the different modelling techniques and the mesh size have on the solutions.

4.1. UD specimens

Tensile tests on unidirectional specimen represent the simplest one in terms of experimental set up and in terms of material layout, hence resulting in an immediate feedback on the reliability of the simulations. The ultimate strength and the stiffness were the main aspect investigated. Since no delaminations were expected only the PB and TIE models were developed, but the results were almost identical. In Figure 4, the comparison between the simulation and the experimental results is reported, focusing also on the effect of the mesh on the ultimate load prediction. The three meshes previously cited led to a good result: only the smallest size mesh slightly underestimated the failure load. Concerning the stiffness, the FE model, after a first part in which they were identical, was lower than in the experiments.



(a) Load-displacement curves of PB model

Fig. 4. Comparisons of the load-displacement curve between the FE model and experiments of a balanced with hole specimen. Different mesh were investigated

4.2. BAL specimens

The balanced specimens test was simulated through the PB and the TIE approach as reported in Figure 5, also comparing three different mesh dimensions. The TIE model was a little stiffer than the PB one, giving slightly better results; also, the mesh affects the loading curve slope, especially for the PB model: the smaller is the size of the mesh, the greater is the stiffness. However, all the results can be considered good because a certain variability can be found even in the experimental data. The COH model (not reported), was developed to take into account delaminations, gave identical results compared with the TIE model. In fact, delaminations did not occur until the final failure in accordance with the experimental findings. Therefore, the adoption of a cohesive approach does not add information or contributions but is only more expensive in terms of computational cost. The resistance of the balance specimen in tensile test was dominated by the 0° plies, because the 90° and the ±45° laminae were unable to carry high load and they were prone to fail due to tensile matrix failure. Particularly the 90° oriented laminae completely broke first, consequently ±45° carried more load but they did not reach failure until fracture. Hence, those laminae show a diffused tensile matrix damage prior to failure, but the fracture occurs only when the maximum stress in the fiber direction is reached. After the 0° plies failed, strong vibrations were induced and the stress wave propagation caused a distributed break of all the other plies. Furthermore, it must be mentioned that ±45° laminae generally behaved in a very nonlinear way, but when inserted in balanced specimen this effect was much reduced and the linear approximation of the model did not worsen the results. Hence the linear approximation for balanced is acceptable in tensile test simulations and good results can be achieved.

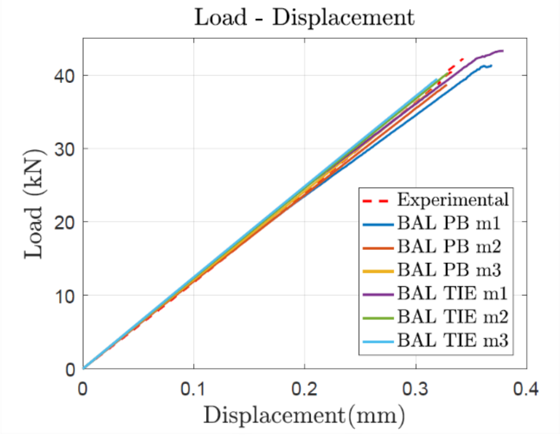


Fig. 5. Comparisons of the load-displacement curve between the FE model and experiments of the balanced specimen: different mesh were considered

4.2.1. Open hole

The open hole test was used to validate the tuned material properties. Also in this case, the PB, TIE and COH models were developed, but no differences were found in the results, see Fig.6. Therefore, the PB model is the most suitable choice to obtain the load-displacement curve, because it is the simplest to implement and also the fastest. In order to be able to analyse the complex stress state near the hole, a fine mesh was used, while close to the grips a coarse mesh was adopted. In between, a transition mesh was used to avoid an unnecessary increase in computational cost. The results of the different models and mesh are reported in Fig.7. The stiffness was always correctly predicted, but the ultimate load was smaller when the mesh dimension decreased. To overcome this issue a mesh regularization technique should be adopted, or as an alternative, a criterion with in-situ strength for a fixed mesh size should be implemented. The mesh regularization method is a methodology that allows to predict the correct ultimate load, modifying the maximum strength value in accordance to the mesh dimension. It is well-known that mesh dimension may affect the results of a simulation and hence some methods have been developed by many authors including Leone (2017), Mikkelsen (2013) and Daniels (2014). Future work has to be performed in this direction and has already been planned.

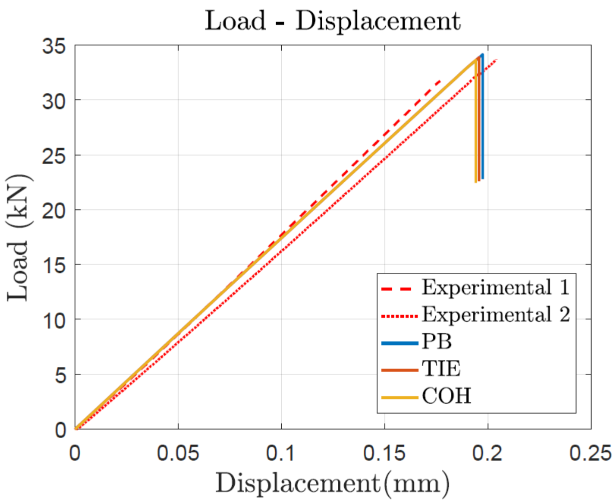


Fig. 6. Comparison of the load-displacement curve between the FE model and experiments of the balanced with hole specimen

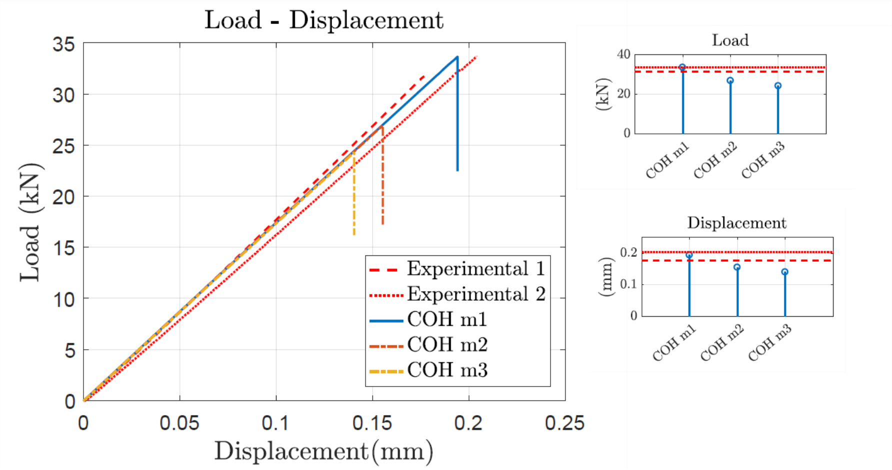
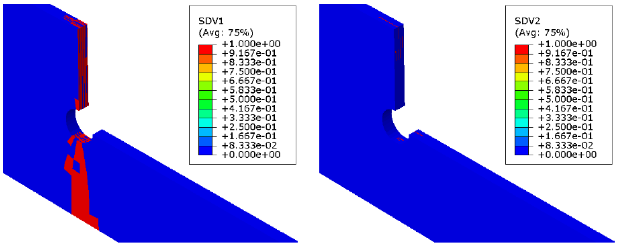
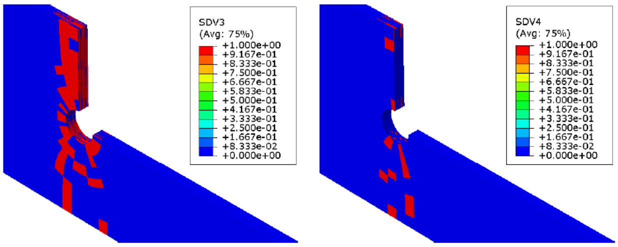


Fig. 7. Comparisons of the load-displacement curve between the FE model and experiments of the balanced with hole specimen: different mesh are used

The three models PB, TIE and OH correctly predicted the stiffness and the failure modes; moreover, the ultimate load was practically the same. As expected the main damage in the specimen was localized near the hole due to a stress intensification as can be seen in Figure 8. The main part of the load was carried by the 0° plies, where failure took place due to the fiber tension and produced a general and instantaneous fracture in all the plies in correspondence of the hole. Damage started mainly in 90° plies around the hole in matrix mode and then propagated into the laminate, while in the ±45° laminae damage tended to propagate along the direction of the fibers. The fiber tensile damage in 0° plies initiated at the tips of the hole and quickly proceeded in the direction perpendicular to the applied load. Fracture occurred due to the matrix and fiber failure in 0° laminae and matrix damage in 90° and ±45° plies, see Fig. 8. The PB and TIE were unable to predict delaminations, which were expected in the area around the hole and were visible in the COH model. Inter-laminar damages started before failure near the hole in such a way that the plies were not separated but the cohesive interaction was degraded. Complete delaminations appeared only when damage evolution was completed (CSDMG = 1), which happened very close to the maximum load, immediately before the final failure, see Fig.9. When delamination occurred, the load transmission between the adjacent plies did not take place and so the breakage of 0° plies did not cause the rupture of the other plies. In fact, 90°and ±45° plies deformed mainly along the matrix direction and hence failed elements were not removed even if they were unable to carry further load. For this reason no laminate breakage into two pieces occurred, only 0° laminae broke. In addition, due to the vibrations after fracture, stress waves propagated in the material and produced other elements failure but the morphology could not be modelled due to the fact that matrix failure does not imply element removal and hence models have some difficulties to exactly determine the fracture shape.



(a) Fiber tensile failure (b) Fiber compressive failure



(c) Matrix tensile failure (d) Matrix compressive failure

Fig. 8. Numerical estimation of the damage close to the hole at failure

Fig. 9. Numerical estimation of the delamination close to the hole. From left to right, the onset of delamination, its evolution and its final extension at the end of the test

5. Conclusions

The purpose of this work was to investigate the response of a FE model aimed to replicate a tensile test of CFRP. Different modelling choices were investigated in order to demonstrate the possibility to apply such framework as a benchmark for fine tuning of the material parameters. The results of the different models are in good agreement with the experimental data also considering all the approximations made, and reflect similar behaviour to the results and the discussions available in the literature. However, despite the numerous studies in the last years, predicting and describing the composites behaviour and their failure with high accuracy is nowadays still a challenge. The composite laminae were modelled as a homogeneous material, investigating its behaviour and damage by the effects at the macro-scale level (lamina level). The numerical simulations were carried out through the ABAQUS\Explicit software developing a quasi-static simulation. To manage the solid elements and the 3D failure criteria, not yet implemented inside the aforementioned software, a VUMAT subroutine, was used. It is worth to note that the material behaviour was considered linear elastic and the Hashin's criterion was used. The so-described approach was applied to simulate a tensile test on UD, plain balanced specimen and balanced specimen with a central hole. The influence of the mesh dimension, constrains and of the interaction type between laminae (PB, TIE, COH) were investigated in detail. Due to the lack of reliable data about the material properties, a calibration of the matrix strength was also carried out. The tuning was performed on the tensile test of the plain balanced specimen and the validation (assessment of the transferability of calibration previously developed) was performed on the specimen with a central hole. The work demonstrates the high sensitivity of the FE model to the numerical choices and in particular in case of the specimen with the central hole the mesh size was shown to play a fundamental role. Hence, it is important to remark that for more complex load cases, in particular when notch (like the hole) are present, a regularization mesh technique or an in-situ strength adjustment could be necessary in order to obtain reliable results. The works further shows that data from the literature have to be carefully considered as small differences in the raw material and in the technological process potentially substantially alter the material behaviour (especially in proximity of the failure). Therefore, it is necessary to perform a material calibration (and validation) and the framework of tensile test of plain, and with a hole specimen, appears to be very adequate to fulfill such a goal.

References

Anonymous, 2013. Abaqus / Explicit Simulation of the Low Velocity Impact of an Aluminum Honeycomb Sandwich Panel. Abaqus Technology Brief

Boria, S., Belingardi, G., 2014. Composite impact attenuator with shell and solid modelling. The 11th World Congress on Computational Mechanics (WCCM XI) and the 5th European Conference on Computational Mechanics (ECCM V), pp. 1–9

Cicco, D. D., Asaee, Z., Taheri F., 2016. Low-velocity impact damage response of fiberglass / magnesium fiber-metal laminates under different size and shape impactors. Mechanics of Advanced Materials and Structures, vol. 0, no. 0, pp. 1–11

Daniels, M., Hyder, I., Suryan, L., Parmigiani, J., 2014. Mesh selection for progressive failure modelling of carbon fiber panels in mode iii shear using an explicit finite element solver. ASME International Mechanical Engineering Congress and Exposition

Feng, D., Aymerich, F., 2013. Damage prediction in composite sandwich panels subjected to low-velocity impact. Composites Part A: Applied Science and Manufacturing, vol. 52, pp. 12–22

Gibson, R.F., 2016. Principles of Composite Material Mechanics. Fourth Edition: Edition 4, CRC Press

Guo, W., Xue P., Yang J., 2013. Nonlinear progressive damage model for composite laminates used for low-velocity impact. Applied Mathematics and Mechanics.

Leone, F., Davila, C. G., Mabson, G., Ramnath, M., Hyder, I., 2017. Fracture-based mesh size requirements for matrix cracks in continuum damage mechanics models. AIAA Structures, Structural Dynamics and Materials Conference

Mikkelsen, L. P., 2013. Mesh dependency of smeared-out non-linear composite finite element models of compressive failure mechanisms in composite materials. XII International Conference on Computational Plasticity, Fundamentals and Applications

Ridgard, C., 2008. Complex structures for manned/unmanned aerial vehicles delivery order 0019: Low temp composite processing mechanical property data. Tech. rep., Advanced Composites Group, Inc.