EXPERIMENTAL AND NUMERICAL INVESTIGATION ON THE CRASH BEHAVIOUR OF COMPOSITE STRUCTURES

F.Veneziani[1*], A.Giubellini[1], L.Vescovi[1], F.Aymerich[2], M.Quaresimin[3]

[1]*Dallara Automobili, via Provinciale 33, Varano de' Melegari, Italy*
[2]*Department of Mechanical Engineering, University of Cagliari, Piazza d'Armi, Cagliari-Italy*
[3] *Department of Management and Engineering, University of Padova, Str. S.Nicola 3, Vicenza-Italy*
**f.veneziani@dallara.it*

Keywords: composite material, crash, finite element analysis

Abstract

This paper presents a study on CFRP structures crash simulation. In particular, the focus is on the implementation of a modeling methodology for applications in automotive and racing fields. The study is conducted on an omega-shaped monolithic specimen. A preliminary static testing phase is performed to characterize the material mechanical properties in tension, compression, and shear; then, the laminate toughness is measured in order to quantify the delamination resistance. The experimental data are used to define plies and interface properties for the FE model representative of the specimen in the numerical simulation of crash testing. Quasi-static and dynamic compressive tests are eventually executed in order to validate the methodology.

1 Introduction

The use of CFRP structures for automotive and racing applications is becoming more and more common as manufacturing technologies improve; composite materials, in fact, allow considerable weight saving which results in higher performances and reduction of environmental pollution. While the study of composite materials in static conditions has reached a certain level of reliability, with a large amount of failure theories available, the engineering of CFRP structures in impact conditions still poses relevant issues, especially when they must be designed for energy absorption purposes, as discussed in [1] and [2]. At the same time, Finite Element Analysis has become a necessary tool for the design of complex structures, since it implies higher analysis accuracy which means time and cost savings otherwise unachievable. For these reasons, the simulation of CFRP structures in a wide range of working conditions is becoming the focus of an increasing number of research and development activities. These considerations form the background for the study presented in this paper, which provides a methodology for the investigation of the energy absorption of CFRP structures by making use of FE simulation techniques.

A current common approach to the analysis of the impact behaviour of CFRP structures is based on a reverse engineering approach, which requires preliminary dynamic tests on specimen similar to the final structure in order to tune the FE model material properties. This fact is a major drawback if FE simulation is used as a predictive tool in the design phase, when the shape and stacking sequence are still to be defined.

This study presents a modeling methodology for the crash analysis of CFRP structures without making use of preliminary dynamic tests on specimens.

2 Description of the structure

The study is performed on an omega-shaped specimen shown in figure 1.

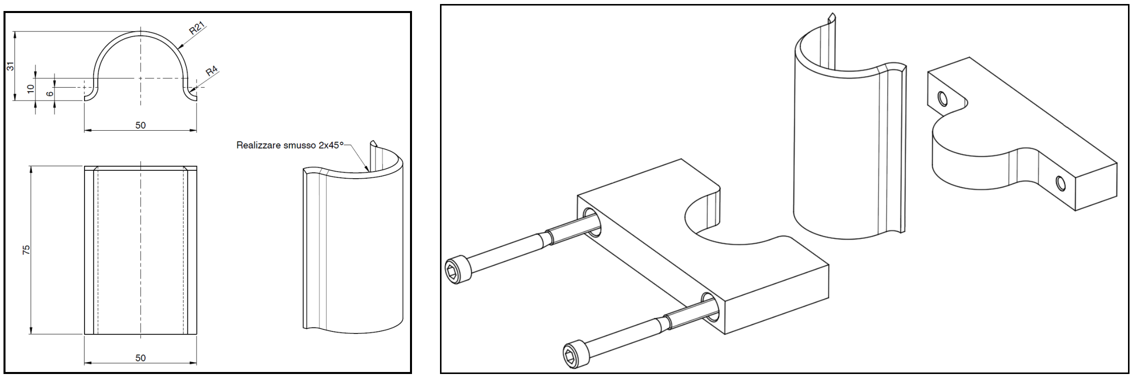

Figure 1. Crush specimen geometry and dimensions

The geometry is designed in order to be stable in crushing conditions, thus providing the ideal background for a comparison between FE simulation and real tests. More detailed information about this kind of geometry can be found in [3]. The lay-up of the specimen is chosen to be simple, in order to limit the number of variables that could have an influence on the crushing performance: monolithic structure with 5 plies (woven T700 carbon fibres/epoxy resin) with the 0° direction parallel to the symmetry axis of the specimen. The total thickness of the specimen is 2.1 mm. On the edge close to the impacting surface, a chamfer is introduced with the function of initiating the stable crushing phase and avoiding possible buckling failure due to initial load peak. The specimen is clamped at the base between two plates screwed together that provide support to the lower part of the specimen (figure 1).

3 Material characterization

Two types of characterization tests are needed.

Static tests: standard tensile/compression tests in 0°/45°/90° direction are performed, according to the rules listed in table 1.

Parameter	Angle	Rule
Tensile stress-strain curve	0°/90°	ASTM D3039
Compression stress-strain curve	0°/90°	ASTM D695
In-plane shear	±45°	ASTM D3518

Table 1. Static tests performed on the woven fabric laminates

From these tests a complete set of stress-strain curves in the relevant directions of the material are obtained, as shown in figure 2. Data presented in the graphs, here and later in the paper, are scaled by a known factor.

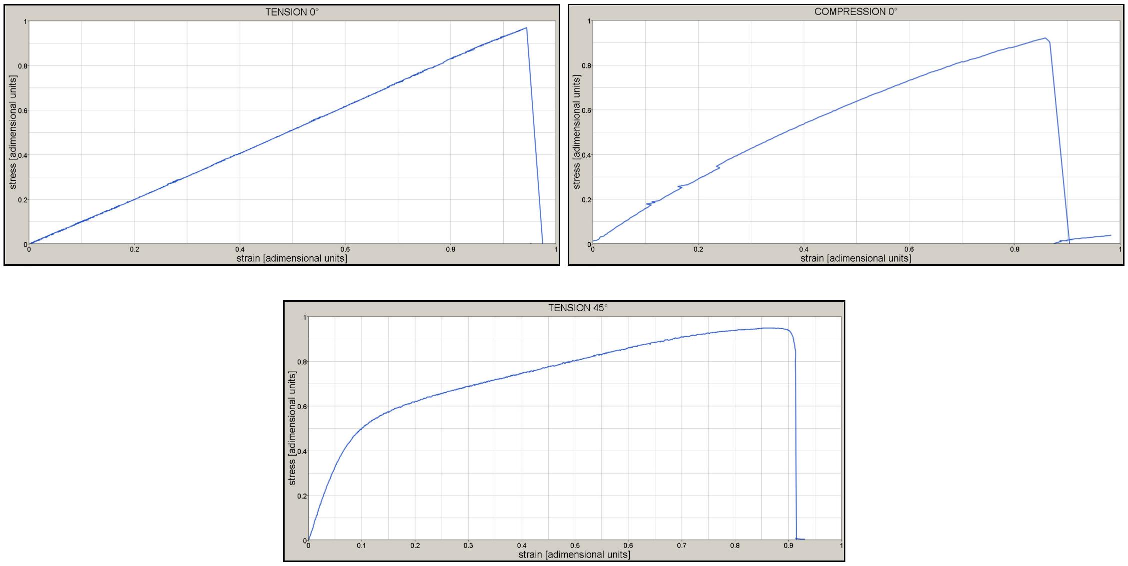

Figure 2. Stress strain curves: tension 0°/90° (top-left), compression 0°/90° (top-right), traction 45° (bottom)

Toughness tests: these tests are required in order to characterize the delamination behaviour of the laminate. Delamination can occur in two different modes: mode I, corresponding to peeling, and mode II, corresponding to sliding.

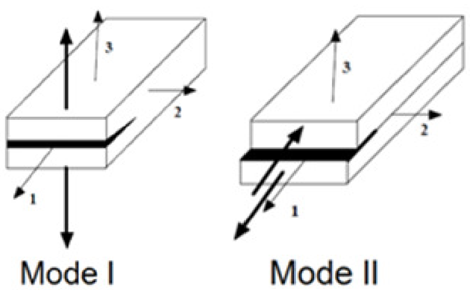

Figure 3. Main delamination modes

A specific test is performed for each mode, besides additional mixed-mode tests which take into account the simultaneous presence of the two modes in different ratios.

Parameter	MM	Test	Rule
Interlaminar toughness mode I	0	DCB	ASTM D5528
Interlaminar toughness mixed mode	0.25	MMB	ASTM D6671
Interlaminar toughness mixed mode	0.5	MMB	ASTM D6671
Interlaminar toughness mixed mode	0.75	MMB	ASTM D6671
Interlaminar toughness mode II	1	ENF	PrEN6030

Table 2. List of toughness tests performed

The mode-mixity (MM) parameter identifies the ratio between mode I and mode II according to the following formula:

$$MM = \frac{G_{II}}{G_I + G_{II}}$$

The mixed mode tests provide additional information on the toughness properties, so that a more precise set up of the FE interface properties is possible.
From the tests, the force-displacement curves are obtained together with the values of critical strain energy release rate for each mode.

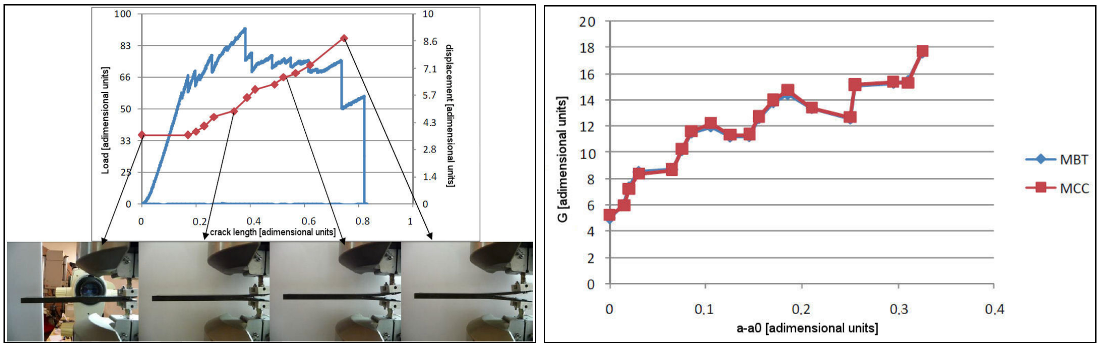

Figure 4. DCB test force-displacement and crack opening-displacement curves (left) and DCB test G_I –crack opening curve (right)

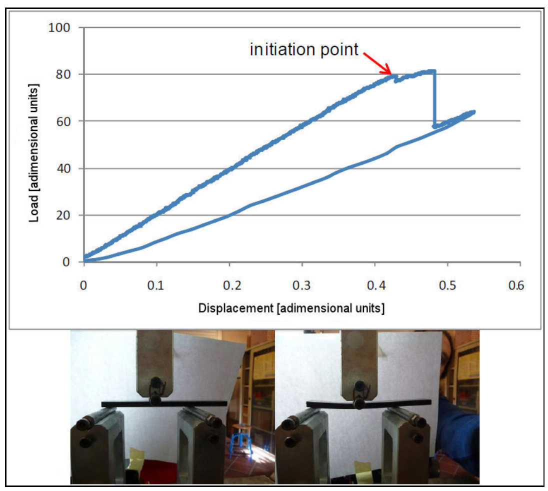

Figure 5. ENF test force-displacement curve

All the values in the graphs in figure 4 and 5 are scaled by a known factor.

4 Finite element dynamic simulation

4.1 Software used

Finite element simulations presented in the paper are performed by using the commercial code Altair Radioss™. It is an optimization-ready, highly parallelizable software which includes a wide variety of tools for the analysis of different types of materials. In particular, several element formulations and failure theories for orthotropic materials are available, making Altair Radioss™ a state-of-the-art software for composite structures analysis.

4.2 Definition of plies and interface properties for FE model

The data measured as described in the previous paragraph are used to characterize the properties for the plies and their connection interface in the FE model. This phase is very important since it defines the in-plane failure modes and delamination characteristics of the FE model; at the end of this phase, the cards of the FE model materials are created. To this purpose, each static and toughness test is reproduced with a dedicated loop of FE analyses.

Static tests: the FE models of the 0°, 90° and 45° tensile and compression tests are build. Then the failure theory which best replicates the behaviour of the plies in all directions is chosen, by tuning the calibration parameters until a good fitting of the stress-displacement curves is obtained.
Toughness tests: the FE models of the DCB (pure mode I) and ENF (pure mode II) tests are built, with brick elements used to simulate the interface between the plies. The force-displacement curves of the real tests are reproduced by tuning the stress-COD relationship for the interface solid elements. The mixed mode tests are used to understand the interaction between mode I and mode II and thus to make a more refined tuning.

4.3 Description of the model

The material cards obtained are used to build the FE model of the specimen; the objective is to replicate as precisely as possible the behaviour of the structure in terms of both force response and crushing mode, i.e. reproduce the physics of the phenomenon.
This task is pursued by modeling the plies with shell elements, bonded together by brick interface elements as shown in figure 6.

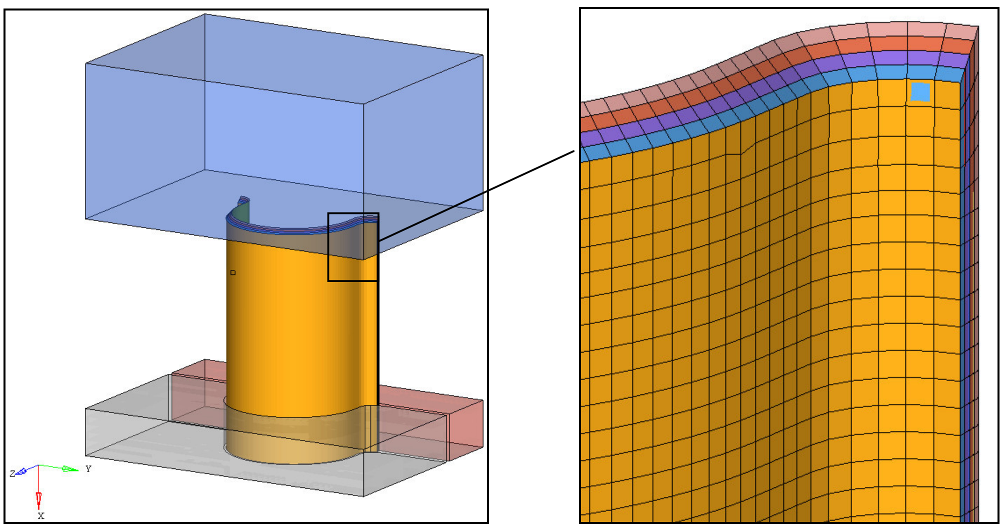

Figure 6. FE model of the crush structure with a detail of the ply-interface solution

This modeling approach allows treating each ply independently, which is what actually happens. During the crushing of a structure, in fact, two different failure modes may occur simultaneously:
Intra-laminar failure: in-plane stresses cause splitting and failure inside each ply
Delamination failure: out-of-plane stresses cause separation between the plies, allowing each ply to behave independently (“petal” shape)
The interaction of these failure modes is very complex and represents one of the main issues in the FE simulation of the phenomenon. For this reason, it has been chosen to create a model as close as possible to the real specimen, modeling each ply and interface separately.
The analysis replicates a drop-tower test, at the following conditions:
Impacting mass=9.57 kg
Speed@impact=6 m/s
Kinetic energy@impact=172 J
The specimen is clamped at the base between two aluminium plates and the mass is forced to move vertically during the impact.

4.4 Simulation results

Figure 7 shows the specimen before and after the impact:

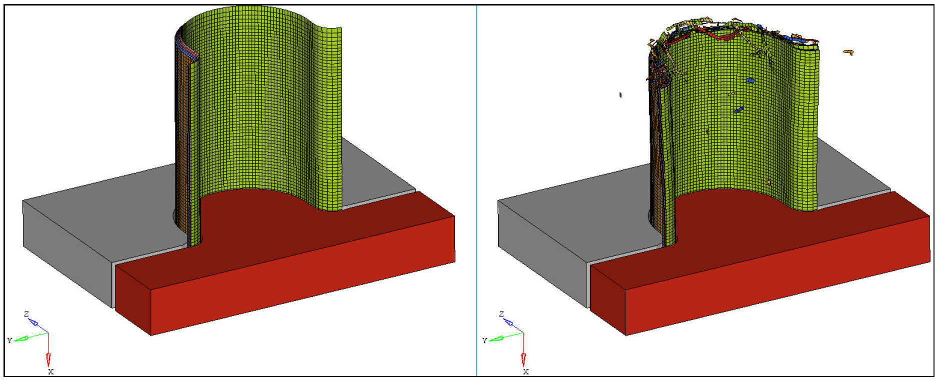

Figure 7. Un-crushed specimen vs specimen after crushing

Figure 8 shows a more detailed view of the specimen edge where crushing is occurring, compared with the same area during the real test:

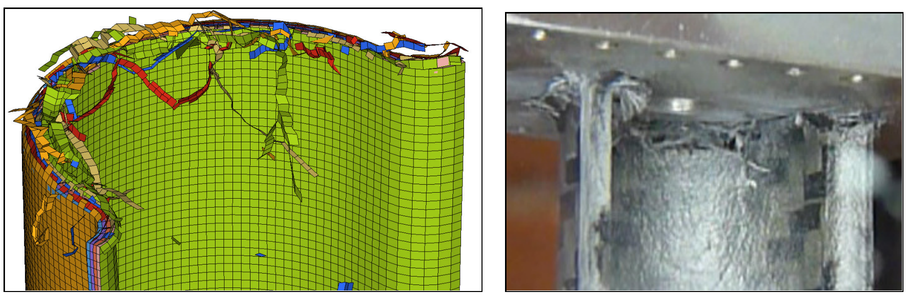

Figure 8. Detail of the crushing edge in the FE model compared with the same area in the real specimen

The similarities in crushing deformation can be noted in terms of delamination and bending of the plies. The force applied by the impactor on the specimen is shown in figure 9.

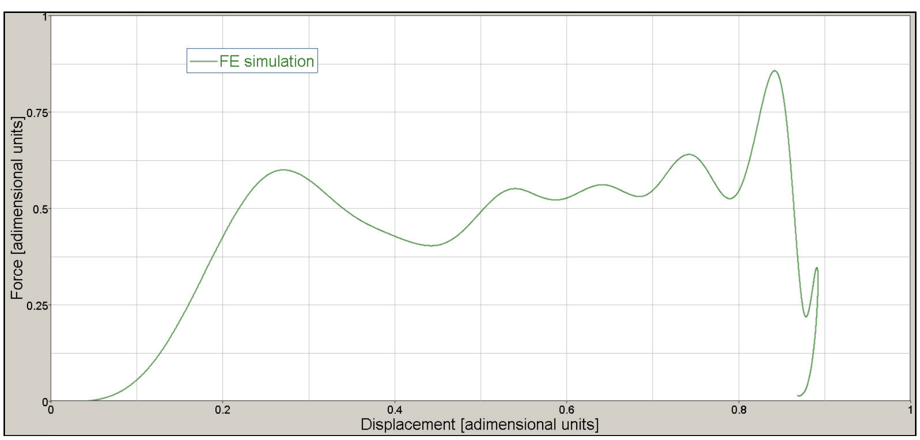

Figure 9. Force-displacement graph of the FE crash analysis

The force and displacement values in the graph in figure 9 are scaled by a known factor.
From the graph it can be noted that the crushing of the specimen is very stable in the first part, with a value of reaction force that remains almost constant.

5 Specimen testing

5.1 Testing conditions

The predictive capability of the proposed FE methodology is evaluated by comparing the simulation output with real tests results. Two test types are performed.

Impact tests: impact tests were conducted using an instrumented drop-weight testing machine equipped with a 9.57 kg impactor with a steel loading plate 80 mm x 80 mm in size, shown in figure 10. The velocity of the impactor immediately before impact, 6 m/s, was obtained by an infra-red sensor, while the contact force was measured by means of a piezoelectric accelerometer rigidly mounted on the impactor. Energy and impactor displacement were calculated as a function of time by integration of the contact force versus time signal. The testing conditions are the same of the FE simulation.

Figure 10. Tooling for the impact tests

Quasi-static tests: the specimen is crushed, under quasi-static conditions, at two different crosshead rates : 5 mm/min and 60 mm/min.

Quasi-static tests are performed on a MTS 809 testing system in order to evaluate possible differences with respect to the crushing of the structure made in dynamic conditions.

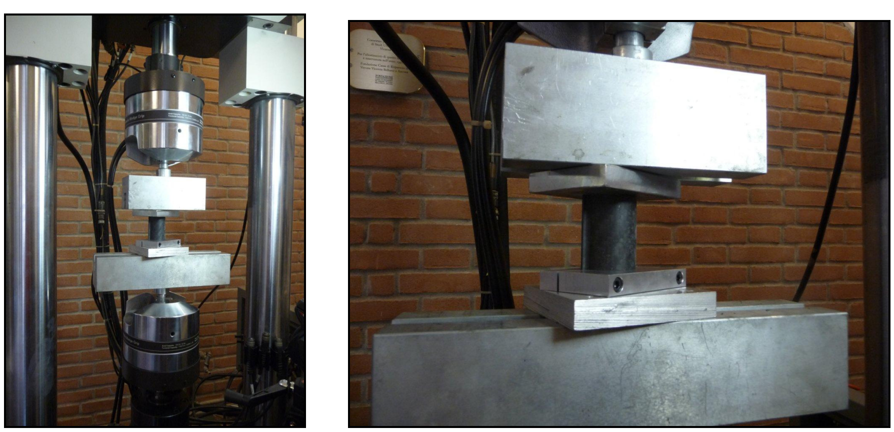

Figure 11. Tooling for the quasi-static tests

The force-displacement graph is plotted to evaluate similarities in the two testing conditions.

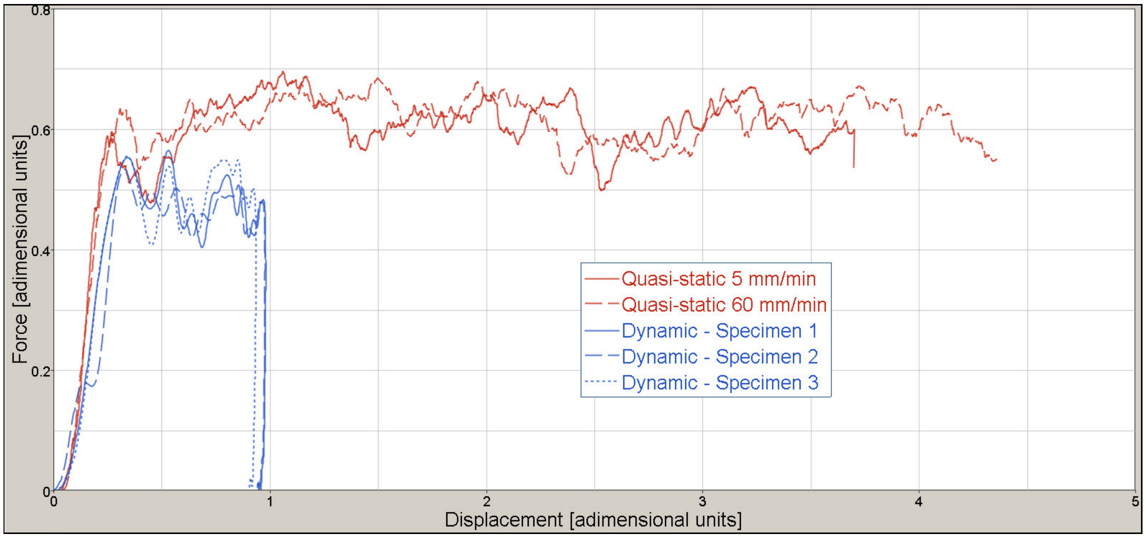

Figure 12. Force-displacement curves for dynamic and quasi-static tests

The graph shows good repeatability among the same test type; on the other hand, a significant difference in the force value is found between quasi-static and dynamic tests.
Finally, the dynamic test data are compared with FE simulation results:

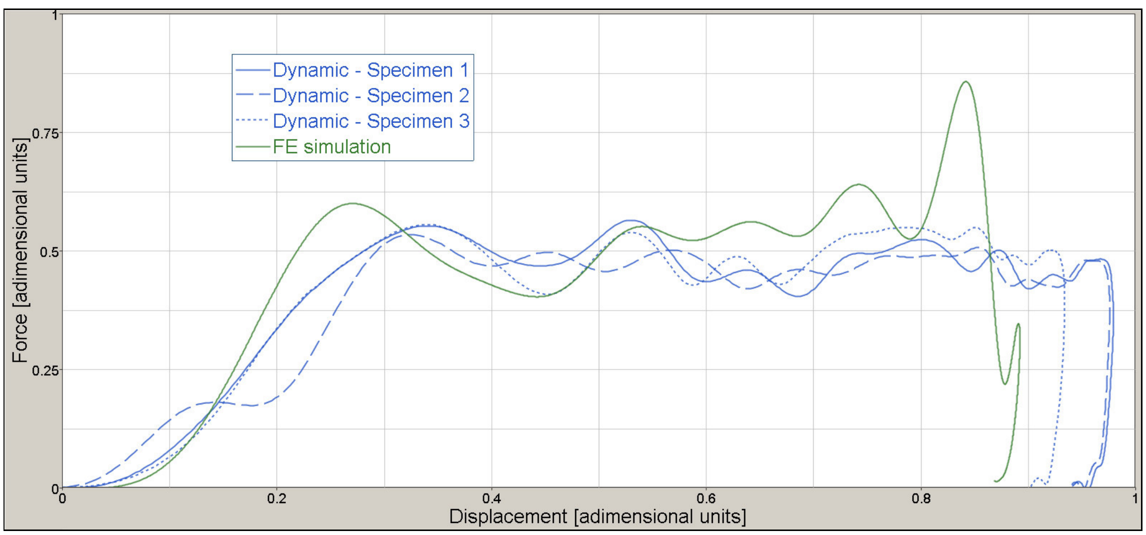

Figure 13. Force-displacement curves of dynamic tests compared with FE simulation results

The graph shows a general good agreement between the tests and FE results, with the initial peak and mean resistant force value being close to each other. The force and displacement values of the graph shown in figure 12 and 13 are scaled by a known factor.

6 Conclusions

The work describes a direct methodology for the FE simulation of the crash behaviour of a composite structure, without the need of a reverse engineering activity: all the material properties for the FE model are obtained from standard static and toughness characterization

tests. The phases leading to creation of the FE model are illustrated and discussed, and the results of a crash simulation of an omega-shaped carbon/epoxy specimen are compared with experimental data from quasi-static and dynamic tests. The predictions of the numerical model are in good agreement with experimental results obtained from impact tests. On the other hand, a crushing force significantly larger than that measured in impact tests was observed under quasi-static conditions. Additional analyses are currently being conducted to identify the main reasons for the different structural response exhibited by the composite samples in the two testing conditions.
Further development of the present work could include the simulation of more complex structures as well as the introduction of different material lay-ups.

References

[1] Reid S.R., Zhou G. (Eds.) *Impact behaviour of fibre-reinforced composite materials and structures*. Woodhead Publishing Limited, Cambridge, UK (2000)
[2] Abrate S. (Ed.) *Impact Engineering of Composite Structures*. Springer, New York (2011)
[3] Johnson A.F., David M. Failure mechanisms in energy-absorbing composite structures. *Philosophical Magazine*, Vol. 90, Nos. 31–32, pp. 4245–4261, 7–14 November 2010