A METHODOLOGY TO ASSESS DAMAGE TOLERANCE OF COMPOSITE STRUCTURES BY FEA SIMULATION TECHNIQUE

Prof. Dr. Uli Burger, Technische Hochschule Ingolstadt, Ingolstadt, Germany
Ludovic Rochat, Institut für Technik und Design GmbH, Ingolstadt, Germany
Clement Breton, Airbus Helicopters, Donauwörth, Germany
Dr. Johannes Markmiller, Airbus Helicopters, Donauwörth, Germany

Abstract

CFRP structures have been widely used since decades in aerospace industry resulting in improvements in payload, fuel consumption and range. The Airbus A350, Boeing B787 in civil airplane industry as well as military products like the NH90 transport helicopter are examples of this development towards “all composite”-aircrafts with nearly all structural parts made of composite materials. In spite of such extensive use, the extent of knowledge of composites remains still low compared to metallic materials. A significant effort in terms of prediction of material response, especially during impact events, is needed to optimize the use of composites. A new time-efficient modelling approach using conventional shell elements is addressed in this paper which allows for an effective evaluation of the damage tolerance of coupons. A methodology based on intermediate tests (DCB and ENF) is proposed to work out relevant material parameters for both damage initiation and evolution of interlaminar and intralaminar modes of failure. The implementation of smart simulations of low velocity impacts and compression after impact (CAI) using Python and Perl scripts, performed to deal with the residual strength of fabric composite materials, is also addressed.

1. INTRODUCTION

Advanced composite materials have been increasingly used in the aerospace industry over the last decade. High specific stiffness, good corrosion and fatigue resistance, crashworthiness and damage tolerance are outstanding mechanical properties that explain the attractiveness of such materials. The recent massive introduction of these lightweight materials in the design of the new generation of aircraft (long-range airliners such as A350, B787, the new civil helicopter of Airbus Helicopters including the military helicopters Tiger and NH90) have contributed significantly to improving critical performances such as the payload, fuel consumption and range. Key materials in today’s design, fatigue and damage tolerance behavior of composites are newly expressly defined in the certification specification (§573 for helicopter composite structures).

Despite a certain trend towards “all-composite” aircraft, the degree of maturity of composite is increasing but still below the one of metals: many investigations, especially in terms of failure modes and progressive damage propagation, remain to be done to optimize the use of composites (i.e. leading to even more lightweight structures). In spite of higher computational power and more sophisticated numerical tools, up to now no general rules are widely accepted to predict efficiently and accurately damage tolerance of composite structures. Complex multi-cracks propagation through matrix and fiber compared to single crack growing for metals explains the difficulties of simulating post-damage behavior within composite lamina. Fracture toughness mechanics is of first importance since events leading to damage are likely to occur frequently throughout the life period of the aircraft, from the manufacturing to the operation and maintenance. Even so-called barely visible impact damage (BVID) are known for decreasing significantly the loading capacity of composite structures [1].

In this paper a methodology is proposed to assess the damage tolerance of composite panel using the commercial FE package Abaqus. Low velocity impact (LVI) and CAI allowing for residual strength assessment of damaged plates as well as simple crack propagation simulations (DCB and ENF) are the focus of the present study. The modelling approach is based on the use of well-known conventional shells along with cohesive elements. The latter provide a way to simulate subsequent laminas separation within a laminate called also delamination. Predominant during LVI, often invisible to naked eye but threatening significantly the integrity of the structure, this mode of failure is of great concern. In the absence of publications of similar studies using the same modeling approach, a procedure is given to perform efficiently sensitivity studies via Python and Perl scripts to adjust numerical parameters, set material properties and transfer numerical results from one simulation to another one. The numerical results are compared with the experiment performed on coupon specimens made of woven fabric layers. Since the ultimate goal of this study is to extend this approach from coupon specimens to real airframe components a special effort is made to keep the computational cost of the simulations at an industrial acceptable level.

For confidentiality reasons, the name of fabric materials used und the value of the material parameters derived in the framework of this project cannot be published. The method itself is extensively discussed.

2. MODELLING APPROACH

So far, experimental, theoretical and numerical methods have been applied to investigate the impact responses on composite structures. From tests, empirical formulae giving limit loads have been established [2, 3]. The theoretical models, such as the spring-mass and energy-balanced one, can give insight into the problem but have their limits [2, 4, 5]. Given the exponential increase in computational power, numerous FE analyses have been performed [6, 7, 8] and good correlation with the experiment have been obtained.

The FE modelling of most of these analyses follows the same pattern: Abaqus model with 3-D elements and cohesive elements in the critical area to model matrix cracks and delamination, stress-based criteria for damage initiation (Hashin or Puck criteria [9, 10] for the in-plane damage, maximum or quadratic stress criteria for the interlaminar damage) and fracture mechanisms for the damage evolution [2, 7, 8]. The explicit solver appears more appropriate even for quasi-static loading due to convergence difficulties of the implicit code when defining contacts and damage evolution [6]. 3-D elements allow for thickness tapering and accurate contact modelling.

Nevertheless, such modelling approaches are computationally expensive, do not capture accurately the bending response and are barely used for lightweight structure modelling in the industry. Thus, in order to make the computational time more acceptable (e.g. max. 1 hour for a coupon specimen of 12 plies impacted at 10J using 4 CPUs) the use of conventional shell elements (S4 in Abaqus) has been investigated.

A rectangular coupon specimen (150x100x2.5mm) made of 12 woven fabric plies has been modelled and impacted at 10 Joules (4kg round head impactor) in Abaqus/Explicit using the following main modelling features (see Figure 1):

- Ply modelling: conventional shell element (S4) have been selected. Since delamination is likely to occur in-between every two adjacent plies (stacking of -45/45 plies), one element through the thickness per ply is used (element size: 1mm).

- Adhesive modelling: 3D cohesive elements COH3D8 modell the adhesion of every adjacent plies to each other (element size: length of 0.5mm, zero geometric thickness, specific thickness of 1mm).

- Intralaminar damage: Abaqus built-in VUMAT subroutine for fabric in-plane fiber breakage and matrix cracking (tension, compression and shear failure criteria).

- Interlaminar damage: the below described “traction-seperation” law capture the delamination onset and propagation.

- Shell – cohesive element interaction: surrounded cohesive elements of any shell element are constrained to each other (same nodal translations) through tie-constraints. Once the cohesive elements are deleted due to severe damage, general contact (hard contact) prevent two consecutive composite plies to penetrate into each other.

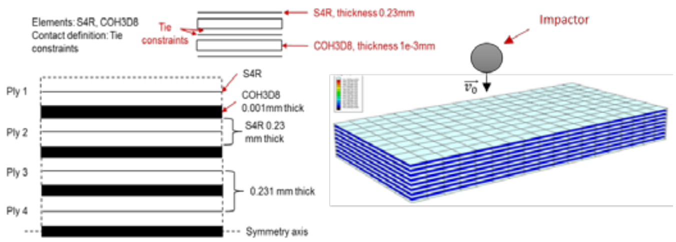


Figure 1: Impact simulation: Modelling approach

The “traction-separation” law available for the cohesive behavior in Abaqus allows for modelling the three modes of delamination onset and propagation, namely opening, shearing and tearing modes (see Figure 2). In order to define the complete cohesive behavior (damage onset and evolution, see Figure 3), three parameters are requested for every mode (i.e. a total of 9 parameters): stiffness E_n, E_s, E_t (slope AB), peak value for nominal stress (shear strength) N, S, T (stress level at point B) and fracture toughness G_{IC}, G_{IIC} and G_{IIIC} (area of triangle ABC) for respectively mode I, II and III. Onset of delamination is based on maximum/quadratic nominal stress/strain criteria. Damage evolution is either energy-based (i.e. fracture energies G_{IC}, G_{IIC}, G_{IIIC} are directly given) or displacement-based (fracture energies are derive from given maximal separation).

The damage evolution for every failure mode is controlled by a coefficient *D* between 0 (at point B) and 1 (at point C). For instance, *D* equal to 0.8 means that the stiffness has been decreased by 80%, i.e. the stiffness is 20% of the undamaged initial stiffness.

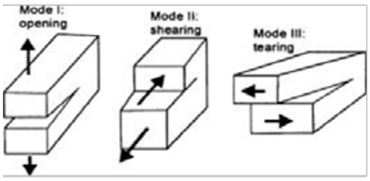


Figure 2: Modes of interlaminar damage

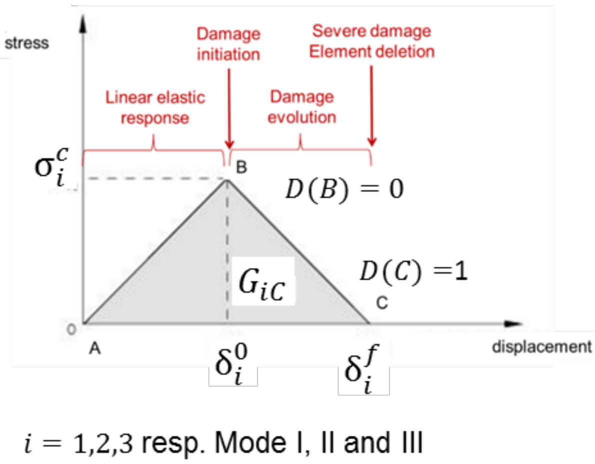


Figure 3: Cohesive behaviour: traction-separation law

3. DERIVATION OF MATERIAL PARAMETERS

Both interlaminar and intralaminar material parameters are hereby discussed. However, only the interlaminar ones (i.e. cohesive parameters) have been extensively investigated. Similar studies remain to be done to define properly the in-plane response.

3.1. Building-block approach

The building-block approach is a method for substantiation of composite structures introduced in CMH-17 [11]. This widely accepted method is based on a fundamental idea: successful completion of complex analyses and tests can only be the product of a thorough work including increasingly complicated intermediate analyses and tests. Such method has been implemented for the derivation of material parameters as shown in Figure 4: one-element models and simple crack simulations have been performed to adjust and validate step by step the material laws and modelling choices used for the impact and CAI simulations.

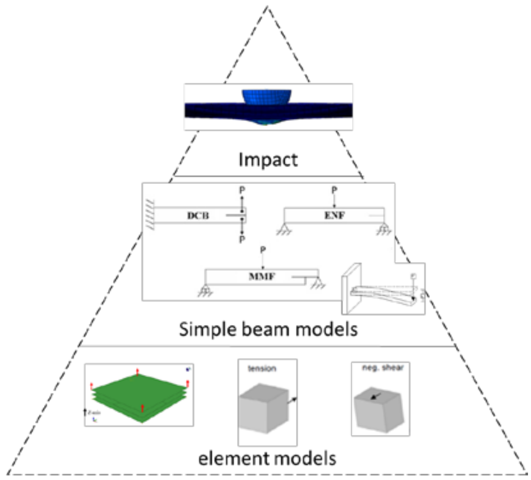


Figure 4: Building-block approach for numerical simulations

The below cantilever beam study is an illustration of the building-block approach. Since the investigated composite plate is submitted to bending during impact, the modelling approach discussed previously (conventional shell elements – cohesive elements and tie constraints) was verified thanks to a cantilever beam (concentrated load at free edge) model, whose boundary conditions and geometry are briefly given below:

- Rectangular plate: 250 x 25mm
- Applied load: 1N
- Lay-up: 4 composite plies (0°) hence 3 cohesive element interfaces

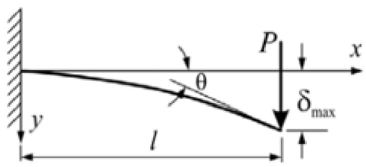


Figure 5: Cantilever beam – concentrated load at the edge

The deflection *y* at any section in terms of *x* is given by the following equation, where *E* and *I* are respectively the Young's Modulus in *x*-direction and the second moment of area of the beam:

$$y = \frac{Px^2}{6EI}(3l - x) \qquad (1)$$

As shown below, the numerical deflection along the *x*-axis obtained via FEA is nearly the same as the theoretical one, thereby contributing to the validation of the modelling approach.

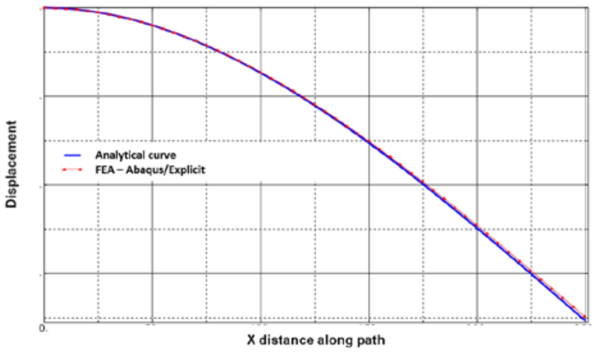


Figure 6: Cantilever beam – concentrated load at the edge

The main objective of this elementary numerical simulation was to investigate the behavior of the tie constraints between the cohesive elements and the conventional shell elements without allowing damage to initiate and propagate.

3.2. Interlaminar fracture

As expected, the numerical results depend significantly on the three cohesive parameters listed previously (stiffness, strength and fracture toughness). However, generally they are not available at the time of simulations (lack of test results) and relevant assumptions have to be made. Article [12] gives tips to overcome such challenges which are leading to satisfactory results for simple single crack simulations (DCB and ENF) but whose relevance regarding more advanced models (such as impact) is limited (according to the present study).

For brevity's sake, the one-element models are not presented due to their simplicity. They are however worth the effort to observe the influence of Abaqus parameters when defining cohesive material laws (such as the Power law or the Benzeggagh-Kenane fracture energy law) [13].

3.2.1. Double Cantilever Beam (DCB)

The DCB specimen are used to investigate the opening mode I of interlaminar failure. The tests are carried out following the standard ASTM-D5528 which proposes an international method to derive the interlaminar fracture toughness G_{IC}. The analytical solution to DCB problems is in the present paper not summarized but for further information one should refer to the extensive literature on this issue [12].

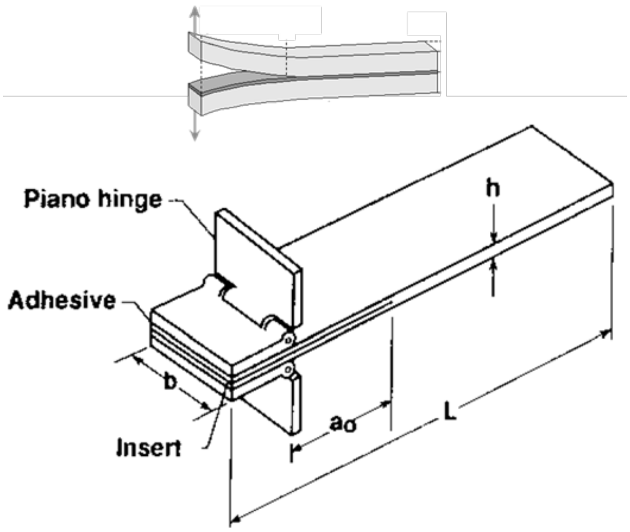


Figure 7: Double Cantilever Beam specimen according to ASTM-D5528

Finite element analyses have been performed and compared to test results, i.e. to crack propagation and force-displacement curves. The modelling approach is the same as the one for the impact simulations (conventional shell and cohesive elements, tie constraints and same mesh density). Knowing the interlaminar fraction toughness G_{IC} from experimental tests, the last two main parameters to define the traction-separation law of the cohesive section (namely the stiffness and nominal stress peak or strength) are adjusted to match the corresponding test results. For instance, in the case below, the stiffness and peak stress have been set resp. to 10 Gpa and 5 MPa and validated for the impact simulations due to good correlation between the DCB theory and test data:

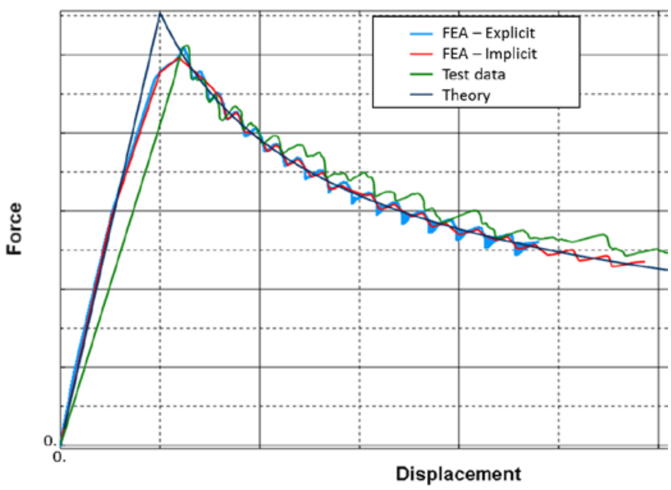


Figure 8: DCB test – force-displacement curves

3.2.2. End Notched Flexure (ENF)

Test for mixed mode I+II has been standardized in ASTM D6671. Regarding fracture characterization in mode II and III separately, no standardized tests exist yet; however, the ENF and ECT (Edge Crack Torsion) tests are due to be approved.

The ENF test consists of a pre-cracked specimen under three point bending loading whose geometry is given below:

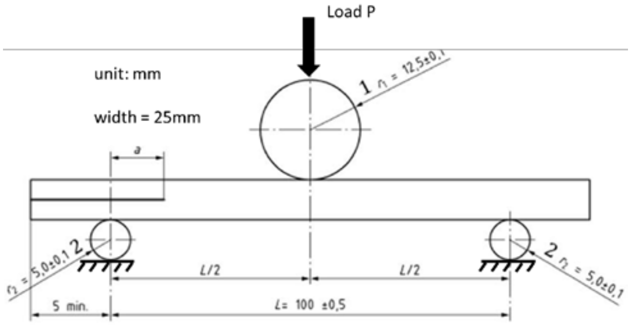


Figure 9: ENF test – geometry

As for the DCB tests, the theory based on energy principles to derive analytically the force-displacement curve and the growing crack length is not detailed in this paper but can be found easily in the abundant literature on the subject [14, 15]. As show in the following figure, the correlation between the theory, the two tests and the FE analyses is satisfactory.

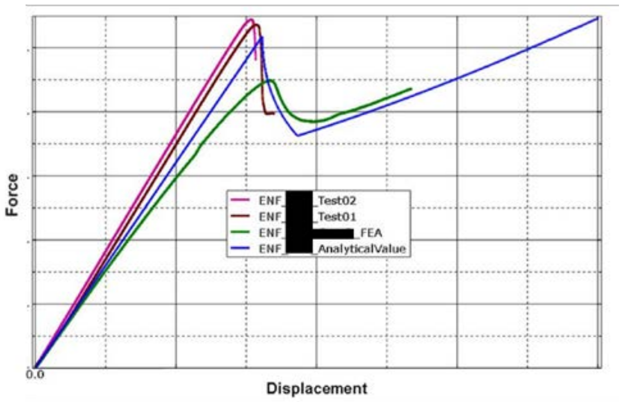


Figure 10: ENF test – force-displacement curves

ENF test is often preferred to calculate G_{IIC} for its simplicity but unstable crack propagation is one of its main drawback. Therefore two other more complex tests might also be standardized, namely the ELS (End Loaded Split, based on the DCB test) and 4ENF (similar to ENF, 4 point bending test allowing for constant bending moment between the two loading noses). These tests have not been performed within the framework of this project.

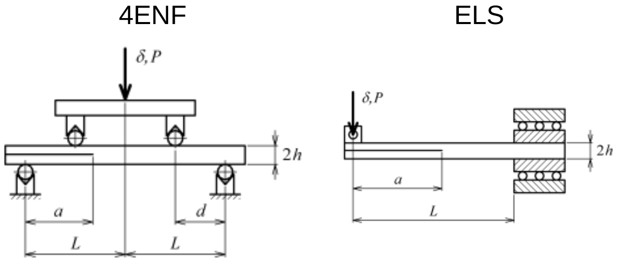


Figure 11: 4ENF and ELS tests

3.2.3. Conclusion: interlaminar parameters

These intermediate tests were definitely useful to derive realistic cohesive parameters. Indeed, the material behavior is more “ductile” than expected. Referring to Figure 3, the displacement δ^f_i (at which the element is completely damaged) was expected to be “close” to δ^0_i (at which the initiation criterion is met), i.e. brittle behavior. However, such behavior leads to unrealistic crack propagation as shown below. The ratio between δ^f_i and δ^0_i is even greater than 30 (see Figure 14).

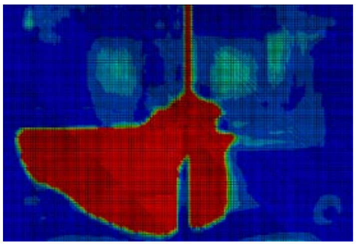


Figure 12: Unrealistic delamination damage propagation (red elements are damaged)

The three intermediate tests DCB, ENF and ILSS (interlaminar shear strength, not discussed in this paper) are used to derive the following parameters (see § 2) for the impact simulations:

- DCB (Mode I): fracture toughness G_{IC} and elasticity E_n
- ENF (Mode II): fracture toughness G_{IIC} and elasticity Es
- ILSS (Mode II): shear strength S
- Mode III = Mode II, no mixed mode (assumption)

Figure 13 is a cross-section of the composite plate and impactor depicting in red the cohesive elements that have been the most damaged ($D>0.9$) when impacted at 10J, i.e. the delamination area through the plate thickness. It has to be noted that, among the several options available in Abaqus, D_{max} has been set to 1 and the element deletion as soon as D_{max} is reached has been activated [13].

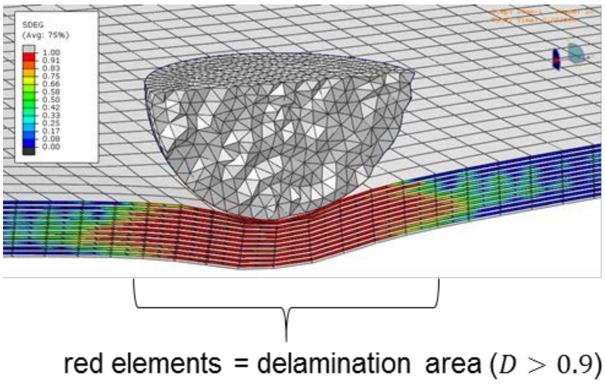


Figure 13: Cross-section of impact model representing through thickness delamination area

For one of these red elements (similar for the other ones), the stress σ_{23}-strain ε_{23} curve corresponding to failure more II is given below (green curve):

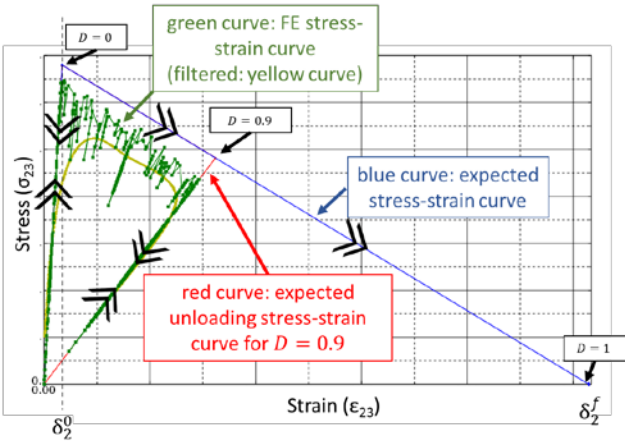


Figure 14: Cohesive material response – stress-strain curve

It can be seen that the red elements are not fully degraded, therefore not deleted (stiffness not equals to 0 but to less about 10% of the initial value). As raised in the §4.1 such delamination area is similar to the experiment. It has to be noted that the ultrasonic devise used to work out the projected delamination area (see Figure 19) records only decreases of material density but cannot accurately indicate if the delamination is partial or total (i.e. partial material degradation or complete separation of two consecutive plies).

If the damaged plate is reloaded (e.g. during compression after impact), the stress-strain behavior will consist of an elastic part corresponding to the red curve (stiffness equals to about 10% of the initial value) and stiffness softening part (damage evolution) corresponding to the remaining blue curve until D reaches 1 (see Figure 14).

3.3. Intralaminar fracture

In Abaqus (like other software packages: LS-Dyna and Radioss), a major difficulty is the mesh dependency associated to the in-plane fracture toughness in both direction (1 and 2) called respectively wrap and weft for woven fabric ply.

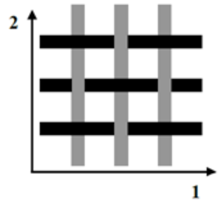


Figure 15: Material direction for woven fabric ply [17]

For impact events up to 10J, the in-plane damage is negligible. However, above 10J fibre breakage failures are clearly visible at both the impacted and non-impacted side of the composite plate. Realistic material parameters would have been necessary to capture accurately the coupon response of such impact events. Nevertheless, the characterization of intralaminar fracture toughness though testing is beyond the scope of this paper. Test procedures are addressed in [15, 16].

A method to derive an approximate fracture toughness of the tensile and compressive failure modes in both warp and weft direction from stiffness and strengths of woven fabric plies is given in Figure 16. The tensile case in the wrap direction (noted 1) is used as an illustration.

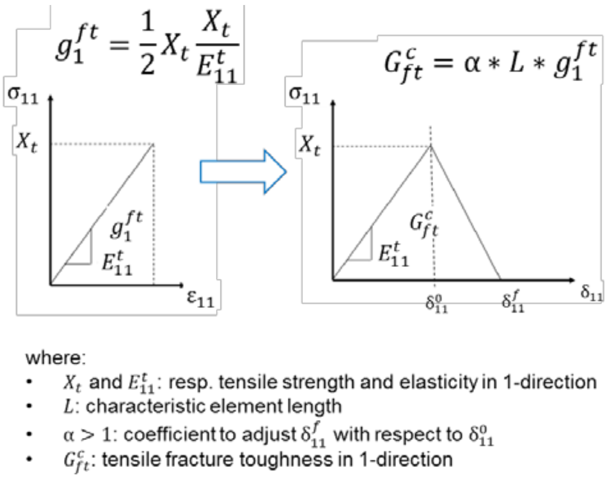


Figure 16: Derivation of intralaminar fracture toughness

The mesh dependency is clearly stated by the equation of Figure 16.

To prevent an over-prediction of the energy dissipation, the following condition has to be fulfilled:

$$G_{ft}^c - L * g_1^{ft} > 0 \qquad (2)$$

Thus, the characteristic element size (square root of the element area for shell elements) of the mesh should not be greater than:

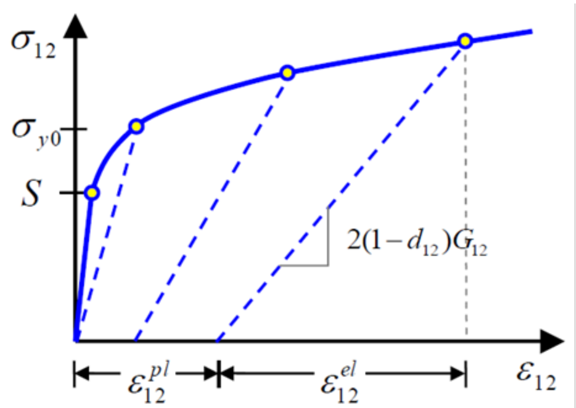
$$L_{max} = \frac{G_{ft}^{c}}{g_{1}^{ft}} \tag{3}$$

It has to be noted that such mesh dependency is not an issue for cohesive element since the solver performs the calculation using a specific thickness (set to 1mm) instead of the geometric one.

Like the cohesive behaviour, the coefficient α was first set up close to 1 to model a brittle behaviour. However, this leads to unstable and unrealistic damage propagation for impact event whose energy is above 10J. More investigation on this coefficient and fracture toughness remain to be done.

Moreover, more testing would have been necessary to get the in-plane strengths of the woven material. The static B-value that have been used might be too conservative to capture accurately the damage due to impact.

Finally, the shear response of woven fabric material should be more deeply investigated. Contrary to unidirectional plies, the fabric shear response should include plasticity. The built-in subroutine of Abaqus for fabric reinforced composites [17] allows to take it into account.



where:

- σ_{12}: shear stress
- S: shear stress at the onset of shear damage
- σ_{y0}: shear yield stress
- ε_{12}^{el}: elastic strain
- ε_{12}^{pl}: plastic strain
- G_{12}: shear modulus
- d_{12}: damage variable for shear

Figure 17: Fabric in-plane shear response [17]

This shear response can be calibrated with cyclic tensile tests on +-45 laminate but such calibration was beyond the scope of this study.

4. LOW-VELOCITY IMPACT SIMULATIONS

4.1. Evaluation of numerical results

In the present paper, the damage tolerance of a coupon specimen is assessed by its residual strength once impacted.

In order to assess the relevance of the numerical impact simulations, five main criteria are systematically taken into consideration: computational time (should be acceptable to be used during projects in the industry, not only during research investigations), the balance energy (total energy should be constant and artificial energy negligible), contact history (realistic contact between the impactor, composite plate and supporting frame should be defined, i.e. prevent penetration), damage area and impact history. The two last criteria are directly compared with the test data.

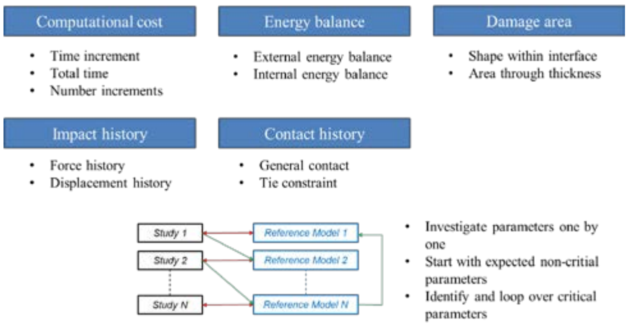


Figure 18: Methodology for validation of FE impact models

For the 10J impact test, good correlation between the numerical and experimental results, in terms of damage area and force history, have been obtained as shown in Figure 19 und Figure 20 using the modelling approach (§2) and the cohesive material parameters (§3).

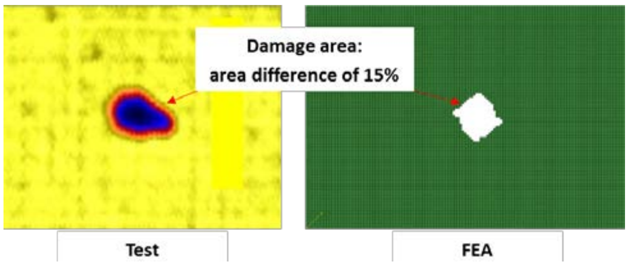


Figure 19: Projected damage area (mainly delamination)

Despite encouraging results for the 10J impact simulations, the investigation has not been extended to other lay-ups and composite material (like UDs) due to time limitation. It has to be noted that for higher impact energies (20J to 40J), the results are still unacceptable. Indeed, the damage propagation is unrealistic, i.e. asymmetric and disproportionate propagation. In the case of an impact energy of 10J, in-plane damage (fiber breakage and matrix cracking) are almost nonexistent compared to impact energy above 20J. Therefore, the poor results for the impact energies between 20J to 40J can be explained by the poor intralaminar material parameters as discussed in §3.3.

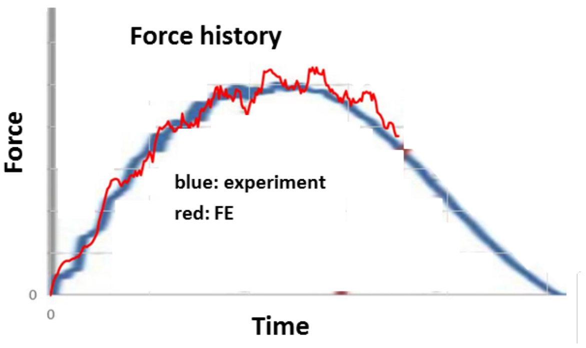


Figure 20: Force applied by the impactor to the composite plate

4.2. Sensitivity studies with Python scripts

Sensitivity studies using Python scripts have been performed to generate quickly a large number of models, almost identical to one another except for a couple of parameters that have been adjusted (material laws, contact definition, mesh density, viscous parameters, cohesive element thickness…) for a better matching of the numerical results with the test data. As one might have expected, it has been found out that the cohesive traction-separation law has the highest influence on the numerical outputs for an impact energy of 10J.

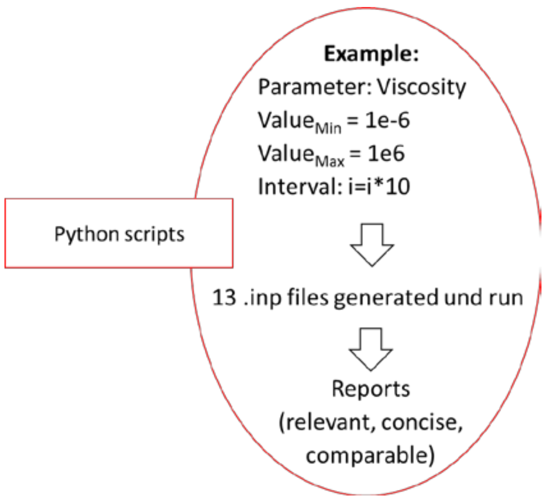


Figure 21: Python scripting

Python scripts are an effective way in Abaqus to perform parametric studies by automating repetitive tasks allowing for easy creation and execution of similar models (in which only few parameters have been changed) and straightforward generation of output database to keep track of the results of these numerous models (see Figure 21).

These sensitivity studies enabled to sort out the parameters and identify effectively those which have a significant influence on the results and therefore which need to be deeply investigated. For instance, thanks to the Python scripts, it has been easily shown that the thickness of the cohesive elements (for values between zero and 10^{-3}mm) is not critical (i.e. no influence on the final results).

The existing literature dealing with Abaqus scripting [18] allows for a quickly implementation of scripts.

5. COMPRESSION AFTER IMPACT (CAI)

In order to determine residual loading capabilities, the damaged coupon is submitted to compression loading called CAI. The results of the impact simulations need to be transferred for the compression.

5.1. Perl scripting

Abaqus offers the possibility to restart the impact simulation in order to carry out the CAI one. Several methods have been tested without clear success:

- Transferring the damaged model from Abaqus/Explicit to Abaqus/Standard using global damping and damage damping
- Transferring the damaged model from Abaqus/Explicit to Abaqus/Standard to reach a static equilibrium back into Abaqus/Explicit to load the model quasi-statically
- Only with Abaqus/Explicit: first step = impact event, second step = to bring the model into quasi-static state (with viscous pressure for instance), third step = model loading in compression quasi-statically

The procedures have been tested but no satisfactory results could be obtained due to difficulties like unexplainable drop in the stable time increment or penetration difficulties. Instead, a simpler, more reliable and computational effective method has been developed: the use of Perl script to identify the elements damaged during the impact simulations and delete them or assign new material properties (to take into account their deteriorated state after impact) in the CAI model prior to running the simulation. Thus, only the needed information, namely the material state, are transferred between the two models to prevent any complication and keep the procedure as simple as possible.

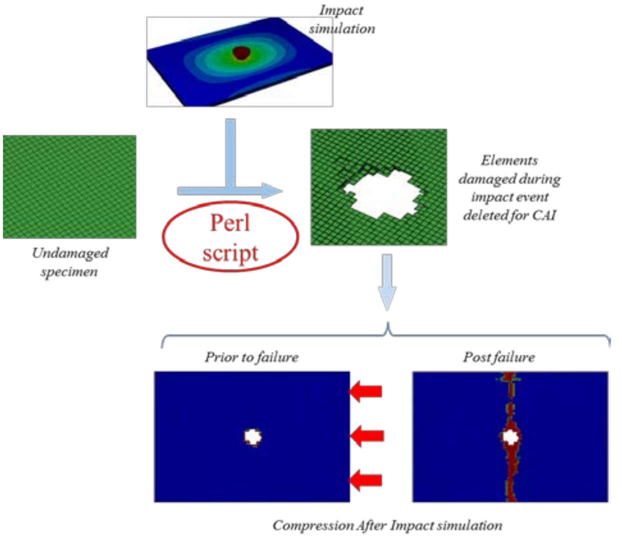


Figure 22: Procedure to assess residual strength of coupon specimen

Since the compression loading in the CAI simulation is quasi-static (15mm/min), Abaqus/Standard would have been more appropriate. However, due to convergence difficulties occurring during the sudden onset of compressive damage, Abaqus/Explicit has been used and the loading speed was increased for computational cost consideration (however not beyond a critical limit at which dynamic effects are no longer negligible). Another or complementary method has been investigated with success to reduce even more the computational cost. The CAI simulation has been run with Abaqus/Standard and stopped slightly before the onset of damage. Then, the model is transferred and restarted in Abaqus/Explicit until final failure of the plate, dividing then by three the computational effort while achieving the same numerical results (as if the model was entirely run with Abaqus/Explicit).

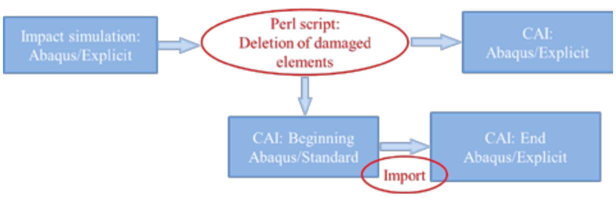


Figure 23: CAI execution via Abaqus/Explicit and Standard

5.2. Residual strength of pre-impacted plate (10J)

The procedure, summarized above, has led to promising results since the difference between the experimental and numerically residual strength for the previously mentioned 10J impact test is less than 5%.

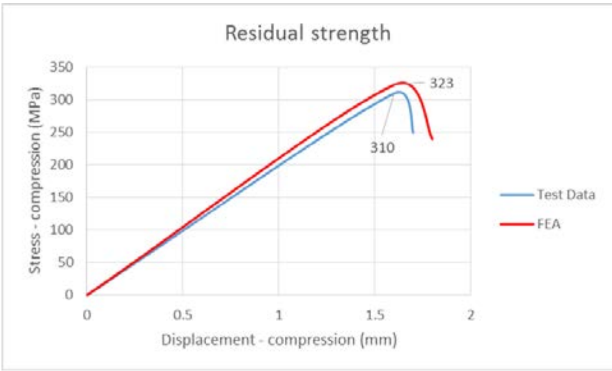


Figure 24: CAI – Residual strength (test and FEA)

The computational time needed to complete the whole loop (impact simulation and CAI of a 100x150mm composite plate made of 12 fabric plies and impacted at 10J) is approximatively two to three hours with four CPUs depending of the loading speed during the CAI simulation (the Perl script runs within a few seconds).

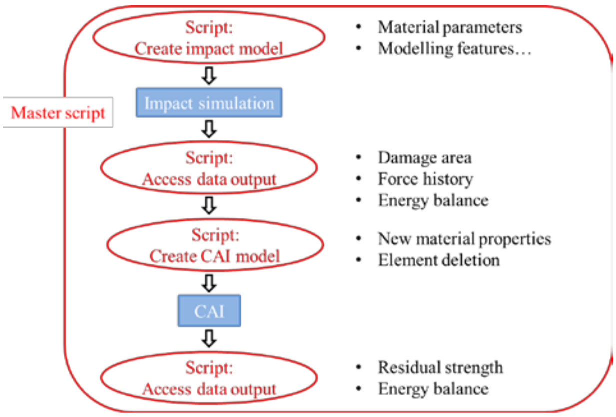


Figure 25: Overview of damage tolerance assessment

6. SUMMARY/CONCLUSIONS

A challenging but promising methodology to calculate numerically the residual strength of impacted coupon specimen is proposed. The modelling approach, based on cohesive element interfaces to capture delamination damage propagation between composites plies modelled by conventional shell elements, is cost effective and has led to satisfactory results validated by the experiment for a particular test: composite plate made of 12 woven fabric plies and impacted at 10J. Python and Perl scripts have been created to make the numerical models more efficient. An appropriate method to derive relevant cohesive parameters has been addressed.

Despite the promising first results, more investigations remain to be done before being able to use such models to predict effectively and accurately the damage tolerance capability of random coupon specimens. For instance, the FE models should be validated with the experiments using different impact energies and composite layouts (ply type and orientation). The present procedure might also be extended to real airframe components like stiffened panel or tail-boom. Deeper investigations to characterize intralaminar fracture remain to be done so that higher impact energy events can be accurately predicted.

7. ACKNOWLEDGEMENT

The present work was performed in the scope of BMWi funded LuFoV-project “Schach” together with Airbus Helicopters. The authors would like to acknowledge the valuable input and efforts by all partners.

8. REFERENCES

1. Engleder, A., Francescatti, D., Burger, U., “A Damage Tolerance and Fatigue Evaluation Approach for Composite Rotorcraft Airframe Structures”, AHS 70th Forum, Montreal, 2014
2. Chai, G.B., Foo, C.C., Seah, L.K., “A model to predict low-velocity impact response and damage in sandwich composites”, Composite Science and Technology, vol 68, pp 1348-1356, 2008
3. Sebaey, T.A., Gonzàlez, E.V., Lopes, C.S., Blanco, N., et al., “Damage resistance and damage tolerance of dispersed CFRP, Design and optimization”, Composite Structures, 95, 569-576, 2013
4. Abrate, S., “Modeling of impact on composite structures”, Composite structures, vol 51, pp 129-138, 2001
5. Olsson R., “Mass criterion for wave controlled impact response of composite plates”, Composites, Part A 31, 879-887, 2000
6. Aymerich, F., Dore, F., Priolo, P., “Simulation of multiple delaminations in impacted cross-ply laminates using a finite element model based on cohesive interface elements”, Composite Science and Technology, 69, 1699-1709, 2009, Italy
7. Wisnom, M.R., “Modelling discrete failures in composites with interface elements”, Composites, Part A, 41, 795-805, 2010, Uk
8. Bouvet, C., Castanié, B., Bizeul, M., Barrau, J.J., “Low velocity impact modelling in laminate composite panels with discrete interface elements”, Inter. Journal of Solids and Struc, 46, 2809-2821, 2009, France
9. Hashin, Z., “Failure Criteria for Unidirectional Fiber Composites”, Journal of Applied Mechanics, vol. 47, pp. 329–334, 1980
10. Puck, A., Schurmann, H., “Failure analysis of FRP laminates by means of physically based phenomenological models”, Comp Sci Technol, 58, 1045-1067, 1998
11. Composite Material Handbook, Volume1: Polymer matrix composites guidelines for characterization of structural materials, CMH-17-1G, March 2012
12. Diehl, T., “Modeling Surface-Bonded Structures with ABAQUS Cohesive Elements: Beam-Type Solutions”, ABAQUS Users Conference, 2005
13. Abaqus Analysis User’s guide 6.14, Elements, Cohesive elements, section 32.5
14. O’Brien, T.K., Johnston, W.M., Toland, G.J., “Mode II Interlaminar fracture toughness and fatigue characterization of a graphite epoxy composite material”, NASA/TM-216838, 2010
15. Sham Prasad, M.S., Venkatesha, C.S., Jayaraju, T., “Experimental methods of determining fracture toughness of fiber reinforced polymer composites under various loading conditions”, Journal of Minerals & Materials Characterization & Engineering, Vol.10, No.13, p.1263-1275, 2011
16. Pinho, S.T., Robinson, P., Iannucci, L. “Fracture toughness of the tensile and compressive fibre failure modes in laminated composites”, Composite Science and Technology 66, p2069-2079, 2006
17. VUMAT for fabric reinforced composites, Simulia
18. Overvelde, J.T.B., “Learn Abaqus script in one hour”, 2010

9. ABBREVIATIONS

ASTM	American Society for Testing and Materials
CAI	Compression After Impact
CFRP	Carbon Fiber Reinforced Plastic
DCB	Double Cantilever Beam
ECT	Edge Crack Torsion
ENF	End Notched Flexure
ILSS	InterLaminar Shear Strength test
LVI	Low Velocity Impact
MMB/MMF	Mixed-Mode Bending/Flexure