Potentials and Limitation of FE-based Simulation Methods for Fibre Composite Structures

Alexander Kling, Luise Kärger, Jens Baaran, Jan Teßmer, Richard Degenhardt

German Aerospace Center (DLR)

Institute of Composite Structures and Adaptive Systems

Braunschweig, Germany

Summary:

Current industrial demands for fibre composite primary structures in the area of aeronautics require innovative, experimentally validated simulation methods and tools, to support a cost and weight efficient design and to reduce their time-to-market.

Reliable application of numerical analysis in upfront design challenges not only verification ("solve the equation right") aspects but also the validation of the numerical methods ("solve the right equations") with reliable experimental investigations. Relevant facets with the focus on validation of numerical methods in the area of buckling and postbuckling of stiffened CFRP panel and impact simulation on sandwich structures will be detailed. Potentials and limitations of the simulation methods will be clearly stressed to allow a dependable assessment of its application in upfront simulations in an early design phase.

Keywords:

Verification, Validation, Buckling, Post-buckling, Impact, Experiments, CFRP, Sandwich

1 Introduction

Prevailing industrial demands in the area of aeronautics with respect to efficient (e.g. costs and weight) design of future fibre composite primary structures require new, innovative and experimentally validated simulation methods. The challenge and factor of success in the practical utilization of numerical methods in upfront simulations is strongly connected to a rigorous restriction of the applicability within the validated domain. The day-to-day user (not the developer) of a numerical simulation tool has to know the borderline created within the validation process and the desired domain of interest.
Fig. 1 displays exemplarily in two dimensions (parameter P1 and P2) validated domains centred by a limited number of physical experiments and the grey shaded decreasing confidence-level with increasing distance. The white areas within the domain of interest represent the critical regions of low confidence levels. To ensure that time and cost intense redesigns are minimized within a “first time right” approach the user has to be aware, which level of confidence within the intersection of validation domain and the domain of interest can be expected.

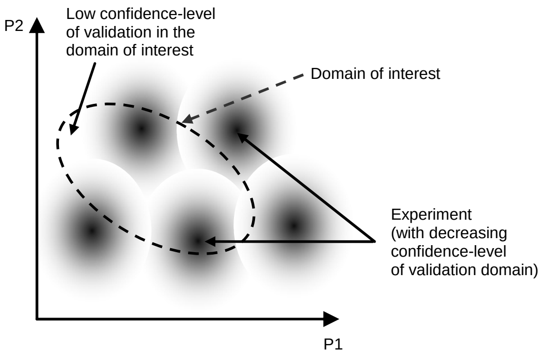

Fig. 1: Visualization: Confidence of validation

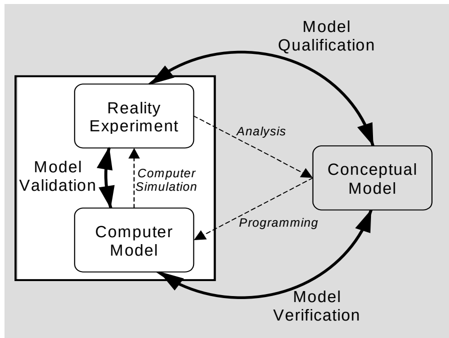

Fig. 2: Phases of modeling and Simulation [1]

The main focus in the subsequently detailed examples is not the verification of the underlying numerical algorithms within the programming phase (e.g. displacement controlled Newton-Raphson Method) and detailed element formulation (“solve the equation right”), but rather the validation aspect (“solve the right equation”) as highlighted in Fig. 2. The phases of modeling and simulation provide an overview and the role of verification and validation in computational analysis. The “Reality” or phenomenological “Experiment” has to be analyzed to obtain the “Conceptual Model” (mathematical equations which describe the physical system). In the following step the mathematical equations are coded to generate the “Computer Model”, which has to be validated with experimental data.

2 Stability Analysis of Stiffened CFRP Panels

The allowable load bearing capacity of undamaged thin-walled stringer stiffened carbon fibre reinforced plastic (CFRP) panels loaded in compression or shear is currently limited by its buckling load. The extension to a novel stability design scenario [2] - to permit postbuckling under ultimate load - requires validated simulation procedures for this highly nonlinear topic up to the deeper postbuckling regime. As restriction, only local buckling and postbuckling with small deformations will be permitted.
Different aspects of the validation process with respect to experiments and their pre-test planning, nonlinear FE analysis as well as the comparison on different levels of detail are highlighted subsequently.

2.1 Experiment

Significant efforts on experiments of stringer stiffened CFRP-panels have been made to obtain a sound database for validation purposes. Due to the time consuming and therefore expensive experiments a substantial amount of work was spent on detailed pre-test analysis and planning. Several nonlinear analyses have been conducted with ABAQUS/Standard to trace the postbuckling behavior. These preliminary numerical studies indicated only a minor imperfection sensitivity. However, additional investigations with respect to boundary conditions along the longitudinal edges of the panels, revealed that the clamping width of the attached longitudinal supports have a significant influence on the postbuckling behavior.
Based on these pre-test analyses a good understanding of the nonlinear behavior of the CFRP test structures has been obtained, which influenced the placement of sensors (e.g. strain gauges) and

examination of critical areas within the experimental investigations. Pre-test planning and numerical investigations is crucial for a reliable and goal oriented validation of numerical results, which finally leads to the concept of validation experiments [3].
Within the finished EU project POSICOSS (Improved Postbuckling Simulation for Design of Fibre Composite Stiffened Fuselage Structures) [4] eight stringer stiffened panels have been manufactured, and tested at the DLR buckling test facility [5]. The results of a four stringer panel will be detailed exemplarily. The blade-shaped stringers have been manufactured separately and subsequently bonded to the skin. Ultrasonic inspections have been conducted to ensure the quality of the panels, especially the bonding of skin and stringer flange. In order to obtain the real shape of the skin, ATOS, an optical 3D digitizing measurement system, was employed to extract the actual radius of the panel as well as the initial geometric imperfections. The locations of the strain gauges were based on the afore mentioned pre-test analysis, however, these have been adapted to cover possibly, slightly different, experimental deformation patterns.
During the experiment, an optical measurement system (ARAMIS) has been utilized to capture digital images of the deformed structure. After postprocessing theses images (88 have been captured during loading), the displacements at nodes of a fine optical mesh representing the surface of the panel have been obtained.

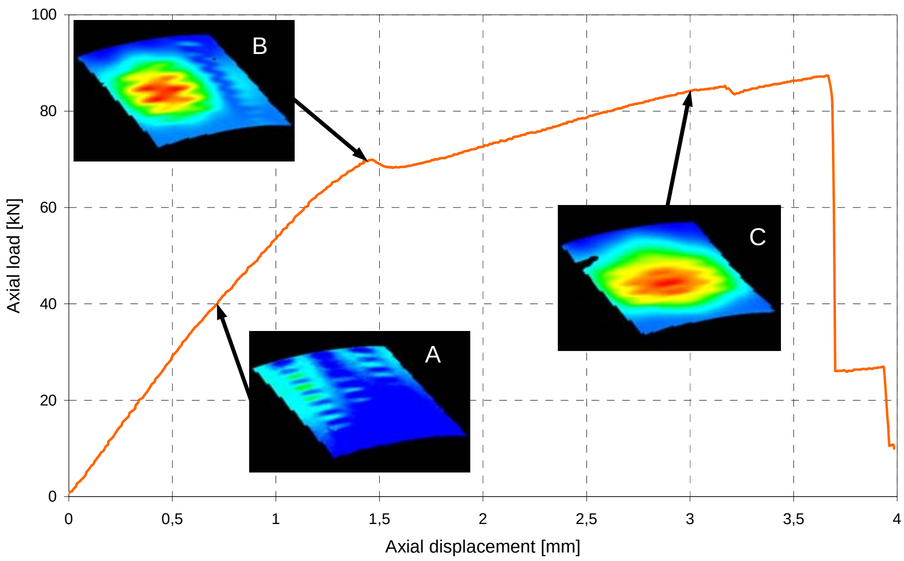

Fig. 3: Load-shortening curve and selected deformation patterns (experimental data)

Three of these displacement patterns (processed color renderings) at characteristic locations are - local skin buckling (A), a 2/3 versus 1/3 global buckle (B) and a single global buckle (C) - along the run of the load-displacement curve are shown in Fig. 3. At an axial displacement of 3.2 mm a small, however sudden change in the load carrying capacity is visible, which is most probably the beginning of local structural degradation.

2.2 Computer Model – Nonlinear Finite Element Analysis

To analyze the pre- and postbuckling behavior of the panels the commercial nonlinear finite element tool ABAQUS/Standard has been employed. A four-node shell element (S4R) [6] has been used to discretize the panel.
Fig. 4 depicts some details of the FE-model (e.g. spring elements, which have been used to introduce the stiffness of the longitudinal edge supports in the computer model).
The approach to conduct the FE-analysis consists basically of four stages (Fig. 5): The preprocessing, a linear eigenvalue analysis to extract buckling modes, which are subsequently used as initial imperfections in the nonlinear analysis utilizing the built-in Newton-Raphson technique with adaptive/artificial damping, and finally the postprocessing. This nonlinear solution method has been proved to be relatively stable for the considered stringer-stiffened panels.
Fig. 6 depicts the load-displacement curves, which have been obtained utilizing the analysis procedure described in Fig. 5 with and without initial geometric imperfections. Comparable to the experimental results displayed in Fig. 3, characteristic deformation patterns, local skin buckling (A), 2/3 versus 1/3 global buckle (B) and single global buckle (C) are shown for the FE analysis without imperfections.

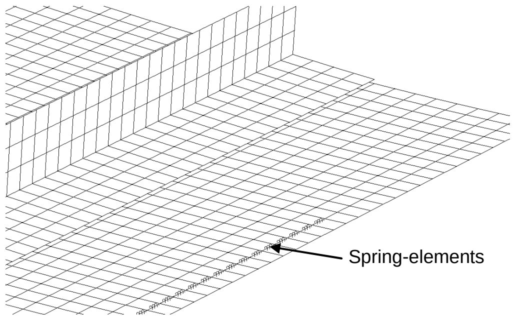

Fig. 4: Details of the FE model

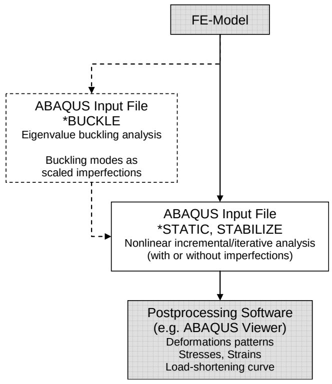

Fig. 5: Analysis procedure in ABAQUS

2.3 Validation

The main focus of this paper is on the so called “global” level of validation, due to the fact that in Fig. 3 and 6 the overall load-shortening as well as the full scale deformation patterns have been compared. The “local” level, where e.g. measurements from strain gauges are considered and compared to numerically calculated strains would have exceeded the scope of the paper.
Fig. 3 and 6 show the good accordance of the numerically extracted and experimentally measured data, like the axial stiffness in the pre- and postbuckling region, the appearance of the non-symmetric global buckle and the global buckle in the deep postbuckling region. Smaller deviations between the experimental and numerical results are visible at the occurrence of the sharp kink (non-symmetric global buckle) and the deeper postbuckling region. This is due to the influence of the rigid supports attached to the longitudinal edges of the panel.

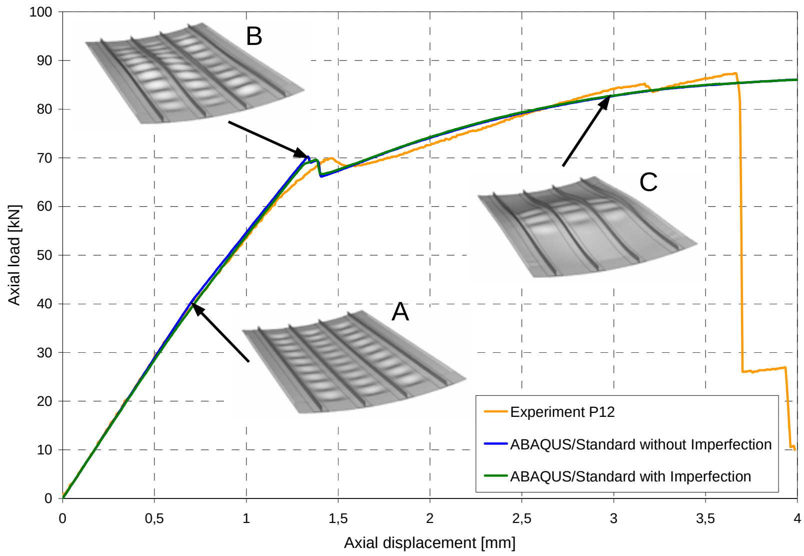

Fig. 6: Load-displacement curve, nonlinear finite element analysis versus experiment

2.4 Limitations

An important issue, especially in the deep postbuckling region up to collapse (e.g. at 3.2 mm axial displacement a small but sudden loss in the load carrying capacity is visible) is the onset of structural degradation, which is not covered within the presented simulation method. However, during the experimental investigations no mayor, detectable damage has been assessed well beyond global buckling. Investigations to consider degradation within the simulation (e.g. skin-stringer separation) and the validation of the improved tools by tests is currently performed within the running EU project COCOMAT (Improved Material Exploitation at Safe Design of Composite Airframe Structures by Accurate Simulation of Collapse) [7], which is under the co-ordination of the Institute of Composite Structures and Adaptive Systems [8].

Another relevant issue is the transferability of the validated numerical results to larger structures, by mainly changing the applied experimental boundary conditions to periodic/symmetric ones, simulating the panel as a subsection of a barrel.

3 Impact Analysis of Composite Sandwich Panels

Composite sandwich panels provide a multi-functional and weight efficient design by the combination of two thin, stiff CFRP face sheets and an intermediate lightweight core. However, impact damage in such composite structures can provoke a significant strength and stability reduction. To improve this situation and to enable an optimal design, the finite element based damage tolerance tool CODAC has been developed for simulating the damage resistance of sandwich structures subjected to low-velocity impacts. Since frequent design loops require a quick simulation, efficient methodologies are needed. Concerning this objective, appropriate models are needed for analysing deformation, failure and dynamic behaviour of sandwich structures. In order to validate the applied impact simulation methodologies, an experimental impact test program acts as referee, where force-time histories and damage sizes are examined.

3.1 Experiments

The impact test programme, which is used to validate the impact simulation methodologies, was conducted at the department of Aerospace Technology at Dresden University (ILR Dresden). Sandwich panels of 400mm x 400mm size were completely supported and laterally fixed against bouncing. The plates were divided into eight parts of 100mm x 200mm, of which seven parts were impacted by energies between 1J and 15J. The impactor had a hemispherical steel tub with a diameter of 25.4mm and a mass of 1.10kg. The studied panels consist of CFRP face sheets (Material CYTEC 977-2/HTA, 0.63mm thick top skin, 2.7mm thick bottom skin) and 28mm thick honeycomb cores of NOMEX 4.8-48.
Fig. 7 exemplarily shows the force-time-histories for impact energies of 1J, 4J and 15J. To illustrate the scatter of the experiments, two curves per impact energy are shown. From the force-time histories two types are distinguishable: The 1J curve is almost sine-like, whereas the 4J and the 15J impacts lead to a sharp bend at about 1kN. This sharp bend is caused by the abrupt failure of the face sheet, which starts to tear at this point. Another much slighter bend is found in all force-time curves at a much lower energy level, when the core starts to fail. These two types of degradation are the key features, which have to be properly considered by the simulation methodologies.

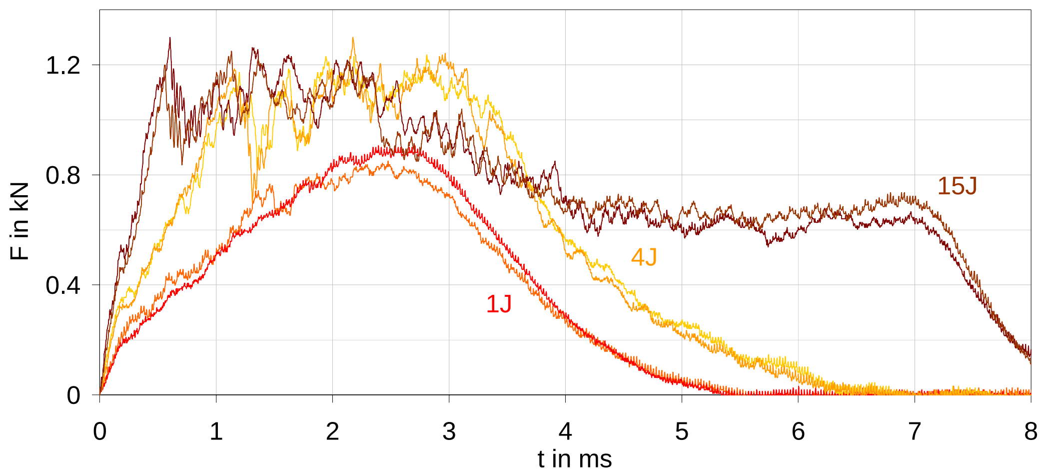

Fig. 7: Force-time histories from impact tests at ILR Dresden

Further important experimental results are damage images. Fig. 8 shows the damage of the top skin and of the core for a 4J impact: At the impact location the skin is indented and torn. The honeycomb core is crushed at an area, which is considerably larger than the skin damage.

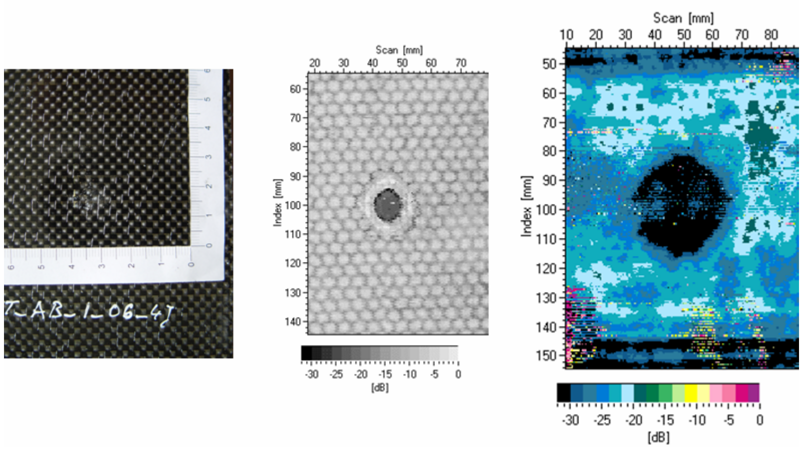

Fig. 8: Damage images from ILR Dresden: top skin photo and ultra sonic scans of back side echoes from top skin (showing damage in impacted skin) and bottom skin (showing core damage area) of a 4J impact.

3.2 Impact simulation methodologies

The dynamic impact process is simulated by using the implicit Newmark time integration scheme [9]. The impactor is modelled by a point mass and the contact between impactor and impacted face sheet is described by the Hertzian contact law. In this regard, Kärger et al. [10] investigated alternative ways of modelling the contact force and showed that distributing the contact force over the contact area is necessary to achieve mesh independent results.
To achieve a rapid and accurate deformation and stress analysis, a three-layered finite shell element S815 is used, which has been proposed by Wetzel et al. [11]. Element S815 is an isoparametric quadrilateral 8-node shell element with 15 degrees of freedom per node (three translations, two rotations for each of the three layers and two further degrees of freedom per layer to describe transverse compressibility). Since an accurate approximation of the transverse stresses is an important requirement for detecting impact damage, transverse stresses are improved by the so-called Extended 2D-Method [12], which has been extended for a three-layered sandwich shell theory by Kärger et al. [13].
To efficiently predict damage growth during the impact event, a progressive damage mechanics approach is applied. For the initiation of core crushing a stress-based failure criterion is used, which includes both, transverse normal as well as transverse shear stresses to detect core failure [14]. If core failure is identified, the material behaviour of the damaged core is modelled according to experimental results of quasi-static tests. Such tests provide not only the material properties of the undamaged core, but also the core crushing stiffness and the core crushing strength. Fig. 9 exemplarily shows force-displacement-curves from transverse compression tests with NOMEX honeycomb cores. The experimental curves are approximated by the stepwise linear function of the core failure model; for details on stiffness reduction factors and core crushing strength see [10].

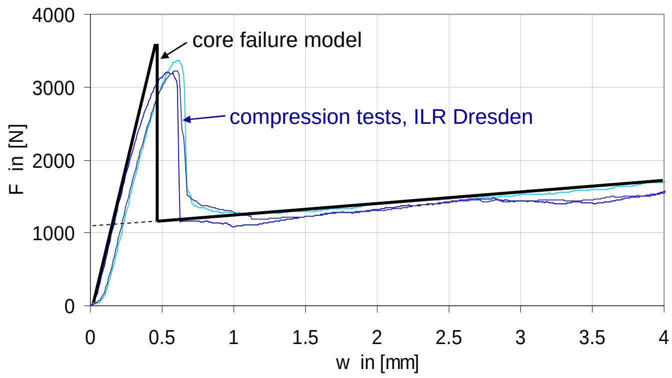

Fig. 9: Force-displacement curves of core compression tests; experimental results by ILR Dresden.

For modelling skin damage during an impact on a sandwich structure, fibre breakage is the most relevant failure mode and needs to be taken into account. Matrix cracking and delamination were found to have insignificant effects on the impact force-time history as well as on the damage propagation. Fibre breakage is predicted by the maximum stress criterion. When the criterion for fibre breakage is exceeded for the first time, the damaged laminas are degraded by reducing the in-plane normal stiffness components according to the results of experimental bending tests. When finally fibre failure occurs in all laminas, the complete skin starts to tear and the force-displacement curve drops down to a very small value. This skin tearing cannot be properly described by the simple linear material models, which are desired for an efficient impact simulation. Consequently, the simulation stops as soon as the skin starts to tear. The objective of the simulation is to properly describe the progressive core and minor skin damage) and to predict the onset of crack growth.

3.3 Validation

The impact energy of 4J shall be taken as an example to compare computational and experimental results in the shape of force-time histories and damage sizes. In this regard, different stages of modelling the material degradation will be discussed. In Fig. 10 three computed force-time histories are compared to experimental curves of two 4J impacts. The top curve illustrates a simulation result, where no degradation is taken into account. This approach clearly overestimates the stiffness of the sandwich plate and computes too large contact forces. By applying the proposed core degradation model, the contact force in Fig. 10 decreases as soon as core damage is detected and a good agreement with the experimental results is found up to a contact force of about 1kN. To simulate the force drop at that point, the skin degradation model needs to be applied. Up to 1kN the skin degradation model causes almost no changes in the force-time curve, since only the two outermost laminas are slightly affected and since the skin is still capable of bearing loads. Face sheet tearing and therewith the substantial force drop does not take place until all laminas have failed.

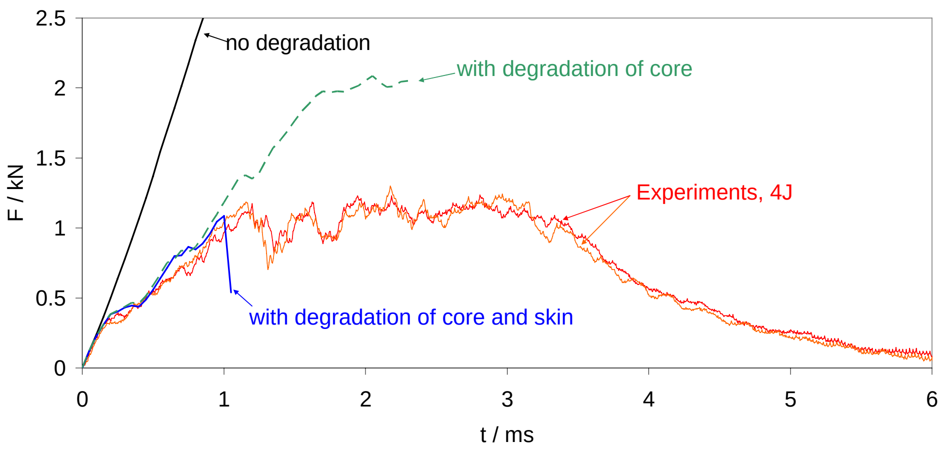

Fig. 10: Force-time histories of a 4J impact for different stages of degradation.

Fig. 11 shows the computed core damage areas for the different stages of degradation (areas with blue crosses) in comparison with the measured core damage area (red quarter circle). If no degradation is taken into account, the core damage is predicted too small, since the stiffness of the sandwich plate is overestimated. In the case of skin degradation the core damage is also too small, since the simulation stops when the skin starts to tear. Of course, in reality the impact process continues at this point, the skin deflects further and the area of core damage increases. Consequently, the computed core damage at the moment of skin tearing is a reliable lower bound of the damage area. A reliable upper bound for the core damage area can be found by a simulation without skin degradation. In this case, the deflection of the intact face sheet is shallower, but affects a larger area. Since the simulation stops at the moment of first skin cracking, the final skin damage size cannot be computed. Hence, the pure presence of a through-thickness skin crack, which firstly and substantially lowers the load capacity of the sandwich plate, is correctly predicted, whereas the final size of the skin crack cannot be predicted.

3.4 Limitations

From the simulation point of view, the limitations of the applied efficient impact methodologies are clearly given by the onset of skin tearing, as described in the two previous sections. However, the pure presence of a through-thickness skin crack, which can be well predicted by the theory and which can be well detected in reality, lowers the load capacity of a sandwich plate already substantially and is, therewith, a key information of impact resistance.

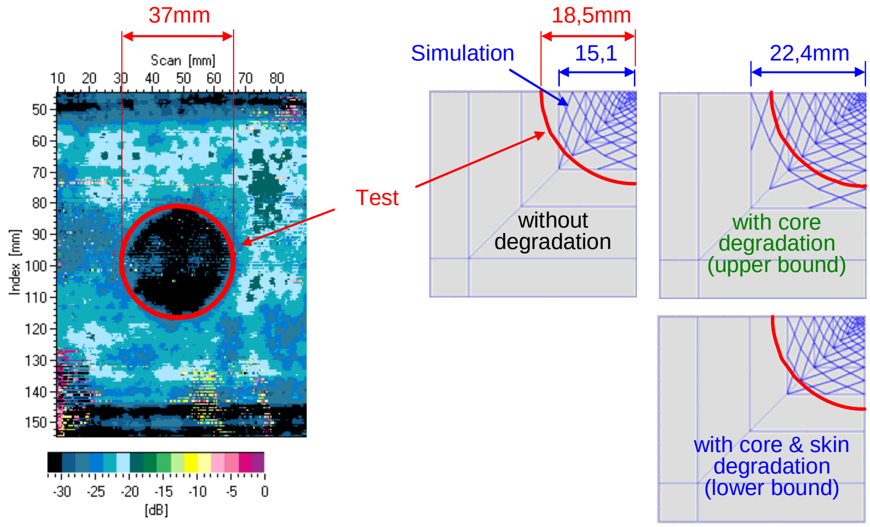

Fig. 11: Sizes of core damage for a 4J impact and different stages of degradation.

Looking at the potentials of validation, there is still a lack of appropriate experimental technologies to detect all possible damage modes in sandwich structures. In most cases, core crushing can be well identified by ultrasonic scans. However, folded core structures with inclined cell walls still cause difficulties and cannot be accessed via US. Moreover, detection of skin damage via non-destructive test methods is strongly limited. While evidence on delamination can be given by the state of the art US technologies, local fibre breakage and matrix cracking cannot be detected via non-destructive techniques. This big drawback obstructs a suitable investigation of the amount of non- or barely visible skin damage. Consequently, particularly the influence of local fibre breakage on the material behaviour cannot be satisfactorily validated.

4 Conclusion

The nonlinear numerical approach to simulate the buckling behaviour of stiffened CFRP panels proved in comparison to experimentally obtained data an acceptable level of confidence up to the deeper postbuckling regime. For efficient and goal oriented validation experiments significant efforts in the area of pre-test planning are crucial, especially for this relatively time and cost intense tests.
A finite element based methodology for efficiently simulating low-velocity impacts on sandwich structures was validated by means of experimental impact tests. Test results showed that impact events of low energy, which induce barely visible damage, can be accurately simulated. Furthermore, it could be validated that the onset of skin tearing and, therewith, the presence of clearly visible impact damage is correctly computed. However, the efficient simulation methodology is not capable of describing crack growth. Furthermore, limitations of non-destructive detection of fibre and matrix cracking obstruct the validation in the regard of skin failure.
Both examples show that a reliable application of numerical methods in upfront simulation require an acceptable level of validation and the knowledge of their individual limitations and assumptions, to minimize cost and time consuming redesigns in a later stage of the development phase.

Acknowledgements

This work was partly supported by the European Commission, Competitive and Sustainable Growth Programme, Contract No. G4RD-CT-1999-00103 (project POSICOSS); the funding is gratefully appreciated. The information in this paper is provided as is and no guarantee or warranty is given that the information is fit for any particular purpose. The user thereof uses the information at its sole risk and liability.

5 References

[1] Roache, PJ. Verification and Validation in Computational Science and Engineering, Hermosa Publisher, 1998.

[2] Zimmermann, R., Rolfes, R.: POSICOSS – improved postbuckling simulation for design of fibre composite stiffened fuselage structures, *Composite Structures*, 73, (2006), pp. 171-174

[3] Oberkampf, W. L.; Trucano, T. G. Verification and Validation in Computational Fluid Dynamics, Progress in Aerospace Science 38, 2002, 209-272.

[4] Zimmermann R., Klein H., Kling A.: Buckling and postbuckling of stringer stiffened fibre composite curved panels – Tests and computations, *Composite Structures*, 73, (2006), pp. 150-161

[5] www.posicoss.de

[6] ABAQUS/Standard and ABAQUS/Explicit, User's Manual, Version 6.4, HIBBIT, KARLSSON & SORENSON, INC.

[7] www.cocomat.de

[8] Degenhardt R., Rolfes R., Zimmermann R., Rohwer K., "COCOMAT - Improved MATerial Exploitation at Safe Design of COmposite Airframe Structures by Accurate Simulation of COllapse", Journal of Composite Structures, Vol. 73 (2006), pp. 175-178

[9] N. M. Newmark: A method for computation of structural dynamics. Journal of Engineering Mechanics - Proc. ASCE 85(EM3): 67-94, 1959.

[10] L. Kärger, J. Baaran and J. Tessmer: Rapid Simulation of Impacts on Composite Sandwich Panels Inducing Barely Visible Damage. Composite Structures, in press, 2006.

[11] A. Wetzel, L. Kärger, R. Rolfes and K. Rohwer: Evaluation of two finite element formulations for a rapid 3D stress analysis of sandwich structures. Computers and Structures 83: 1537-1545, 2005.

[12] R. Rolfes and K. Rohwer: Improved transverse shear stresses in composite finite elements based on first order shear deformation theory. International Journal for Numerical Methods in Engineering 40: 51-60, 1997.

[13] L. Kärger, A. Wetzel, R. Rolfes and K. Rohwer: A three-layered sandwich element with improved transverse shear stiffness and stresses based on FSDT. Computers and Structures 84: 843-854, 2006.

[14] T. Besant, G.A.O. Davies and D. Hitchings: Finite element modelling of low velocity impact of composite sandwich panels. Composites Part A 32: 1189-1196, 2001.