Predictive simulations of damage propagation in laminated composite materials and structures with SAMCEF

Bruyneel M.[1], Naito T.[2], Urushiyama Y.[2], MacDougall S.[3]
[1]SAMTECH s.a. (A Siemens Company) Belgium, [2]Honda R&D Japan, [3]Siemens PLM US

Abstract

In this paper, the advanced damage analysis of composite materials and structures made of continuous fibers embedded in a polymer matrix is addressed. The solution is based on the SAMCEF finite element code, which is now available in the SIEMENS NX CAE environment, with the specific focus of solving non-linear analysis problems for composites. Globally speaking, SAMCEF is an implicit non-linear solver able to solve quasi-static and dynamic problems, with a comprehensive library of structural elements and kinematic joints.

First, the sizing strategy based on the building block approach (pyramid of physical and virtual tests) is recalled. Applied for years in the aerospace industry, it is here extended to the automotive context. In this approach, the knowledge on the composite material and structure is built step by step from the coupon level up to the final full scale structure. In this paper, stages of the pyramid starting from the coupon level are considered, and the predictions obtained by numerical simulations are validated by test results.

The non-linear analysis approach available in the SAMCEF finite element code is then described. It is based on the continuum damage mechanics, and is used to study the progressive failure of composites in the plies and at their interface (delamination). The material models are described. The identification procedure for these damage models is also discussed: it is based on a very limited number of tests results at the coupon level. It is then shown how this information on the material behavior can be used at upper stages of the building block approach and so applied to larger scale structures and/or more complex load cases and different stacking sequences.

The very good agreement obtained in this paper between simulation and test results on composite structures of increasing complexity tend to demonstrate that SAMCEF can be used as a predictive numerical tool for the evaluation of the non-linear behavior of composites, including the progressive inter- and intra-laminar damage analysis.

Introduction

Composite materials have been used successfully in the aerospace industry for many years due to their light weight and high mechanical performances. At the opposite, the amount of carbon fiber reinforced plastics (CFRP) used in the automotive industry is still limited to very specific applications and still not really appears as a reliable solution as far as structural heavily loaded components are concerned. However, vehicle manufacturers and tier suppliers are facing the challenge of consistently maintaining high quality end-product with safety constraints while designing lightweight structures with fuel economy concerns. Carbon fiber-reinforced plastics, because of their high strength to density ratio, represent a serious alternative to classical metallic approach but generate the need to completely redefine the design and sizing methodology of the structural parts. Indeed, composites exhibit complex material behaviors, especially when the assumption of linearity cannot be done anymore. Moreover, composite materials and structures have complex failure modes, which must be well controlled in the sizing process. In this context, predictive simulation tools can be a helpful companion to the physical tests.

In order to propose predictive simulation tools, it is important to use material models able to represent the different modes of degradation of the plies forming the laminated composite structure. Although delamination is a very important mode of failure, intra-laminar failure modes can't be ignored. Inter- and intra-laminar damage modes are studied in this paper, and progressive damages impacting delamination, matrix cracking, fibers breaking, and de-cohesion between fibers and matrix are considered.

Even if lots of models are available in the literature [1-6], the formulation developed in SAMCEF for modeling the damages inside the unidirectional plies of a laminate is based on the continuum damage mechanics approach initially developed in [7], in which the laminate is made of homogenous plies (of various orientations) and damage variables impacting the stiffness of each ply are associated to the different failure modes representing the fiber breaking, matrix cracking and de-cohesion between fibers and matrix. The advantage of this progressive damage model compared to some others is that a parameter identification procedure can be developed. This procedure is based on test results at the coupon level, and allows determining not only the elastic properties but also the value of the parameters describing the non-linear behavior of the material. In this paper, the damage model is first presented, and then the parameter identification procedure is discussed. The parameter values are validated based on a comparison between test and simulation results on a coupon with a stacking sequence that was not used for the identification.

Although different modeling and analysis approaches exist in the literature and in commercial software for modeling delamination [8-13], the cohesive element formulation and relevant associated damage models are here considered [14]. The approach is based on continuum damage mechanics and was initially developed in [15]. The damage model is assigned to some interface elements inserted between the plies to represent their possible de-cohesion and a

fracture criterion is used to decide on the inter-laminar crack propagation. Using such cohesive elements in the analysis allows estimating not only the propagation load but also to predict the failure load, the crack propagation path and the residual stiffness during the fracture process in an automatic way. With this information more accurate safety margins can be assessed. The basics of the parameter identification procedure of such a material model will be briefly explained: test results at the coupon level on DCB and ENF specimens are used to identify the parameters of the damage law.

Once validated at the coupon level, the damage models (for inter and intra-laminar damages) can be used at the upper stages of the pyramid in order to determine the non-linear behavior of larger components and/or more complex load cases and different stacking sequences, where now predictive simulations are the companions of physical tests. Even if the dynamic effects can be treated by the proposed numerical solution, quasi-static tests only are considered in this paper. The material models presented here for inter- and intra-laminar failures are comprehensively implemented in SAMCEF and there is no need for additional plug-ins to solve the progressive damage problem.

The sizing of composite structures

The structures and materials considered in this paper are thin-walled structures made of plies with continuous unidirectional fibers embedded in a polymer matrix. Such composite materials are extensively used in the primary structures of aircrafts. The design of structural composites for advanced applications is nowadays conducted with computers and numerical tools. As explained in [16], it classically involves two disciplines. The first one, called Computer Aided Design (CAD), aims at defining the overall geometry of the part, and the regions of laminates with their stacking sequence. It is linked to the Computer Aided Manufacturing (CAM) and provides specific capabilities for the manufacturing processes simulation. Such capabilities are used to determine the accurate fibers orientations and the deformation of the plies during the draping. At that stage, software like Fibersim can be used. The second discipline, called Computer Aided Engineering (CAE), is used to analyze the structural integrity of the composite structure when it is subjected to the expected loads. In this paper, we only address CAE. It is well know from the aerospace industry that composite structures are sized based on the building block approach [17]. This methodology is described in Figure 1, with the pyramid concept. The idea is to build the knowledge on the material and structural behaviors step by step, starting from the fundamental stage at the coupon level up to the full scale (i.e. the full wing or even the full aircraft). It has been observed over the years that simulation, and especially models based on the finite element method, are more and more used on the different stages of the pyramid, trying to become a companion of the physical tests. It is indeed evident that tests can be expensive when repeated several times for different material configurations (e.g. different stacking sequences) or when changes in the components geometry or loading are studied, and so using virtual testing can help reduce the product development costs. To fulfill this requirement, finite element analyses must be predictive. If this condition is satisfied, simulation can then replace some physical tests.

Developing predictive simulation tools is clearly a challenge. The simulation tools should be able to address different attributes, covering static or quasi-static analyses, damage analyses, fatigue, dynamic response, crash, NVH, etc.

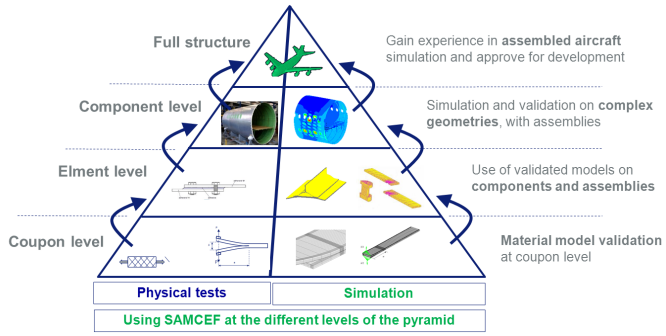

Figure 1. The building block approach applied to aerospace composite structures.

The introduction of effective composite structures in primary parts of automotive vehicles should rely on the approach described in Figure 2, in which the first stages of the pyramids are identical to the ones of Figure 1, and specific applications only appear at the upper stages of the sizing process. The analyst of the automotive industry is therefore confronted to the same problems as the analyst in the aerospace sector: he also needs predictive simulation tools, for the attributes mentioned previously [18].

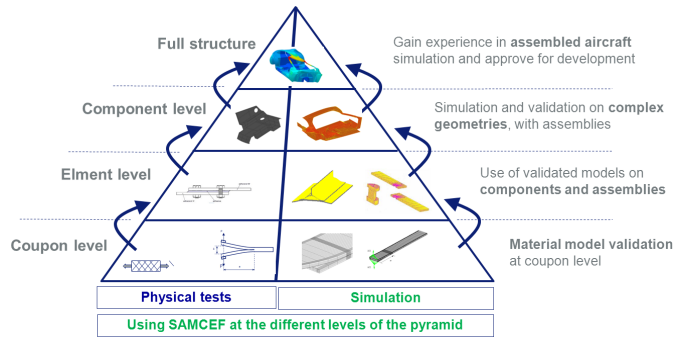

Figure 2. The building block approach applied to automotive composite structures.

Need for a damage tolerant approach

When a laminated composite structure is submitted to a low energy impact, damage may appear inside the structure, especially between the plies. The main issue is that, depending on the energy of the impact, this damage is sometimes not visible (Figure 3). Such damages can actually also appear during the handling of the composite part, or as a result of the manufacturing process. This implies that, in order to avoid overdesigns and not neglect the real behavior of composite materials, composite structures must be sized with a damage tolerant approach, allowing the presence of damage or assuming that damage may be present in the structure even when not visible, in order to determine safe and tight material allowable for the upper stages of the pyramid.

There are of course several ways to address damage with numerical methods. In this paper, the formulation doesn't rely on a multi-scale approach but is based on the continuum damage mechanics and meso-models of the homogenized plies and of the interface are used, which represent the lowest scale in the modelling (Figure 4). However, the physics represented by these meso-models come from a detailed observation of micro, meso and macroscopic behaviors of

the composite material. The approach can be used to study large scale components, well beyond the coupon level [19-21].

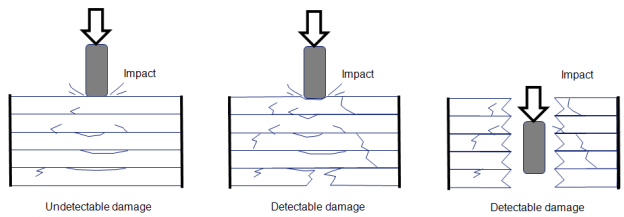

Figure 3. Illustration of the damage level depending on the energy impact.

Modeling inter-laminar damage

Delamination is one of the most critical causes of failure in a laminated composite structure. It results in the separation of two adjacent plies, leading to the propagation of an inter-laminar crack. In the finite element method, the cohesive elements approach is often used to model such cracks (Figure 4), and it is the case in this work. Interface elements are then defined between the finite elements representing the plies. A specific material law with a softening behavior is then assigned to this interface. This allows modeling imperfect interfaces, which are interfaces where delamination can appear in case of excessive loading.

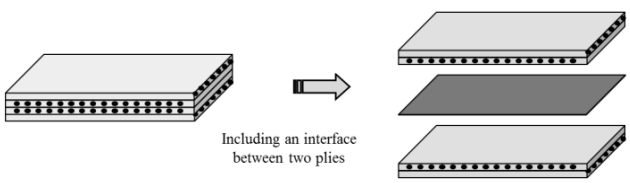

Figure 4. An interface defined in the laminated structure.

In SAMCEF, a potential (that is the energy) including the relevant components of the strain tensor as described in Figure 5 is assigned to the interface elements. In (1), three damage variables d_I, d_{II} and d_{III} , related to modes I, II and III, are defined [15]:

$$e_d = \frac{1}{2}\begin{bmatrix} k_I^0 \langle \varepsilon_{33} \rangle_-^2 + k_I^0 (1-d_I) \langle \varepsilon_{33} \rangle_+^2 \\ + k_{II}^0 (1-d_{II}) \gamma_{31}^2 + k_{III}^0 (1-d_{III}) \gamma_{32}^2 \end{bmatrix} \qquad (1)$$

k_i^0 in (1) are the undamaged stiffnesses.

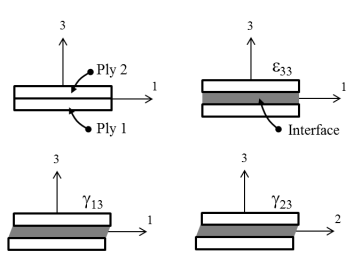

Figure 5. Definition of the interface and inter-laminar cracking modes.

The thermodynamic forces Y_i (i=I,II,III) are obtained by deriving (1) with respect to d_i. For mixed mode loading, the damage evolution is related to the three inter-laminar fracture toughness G_{IC}, G_{IIC} and G_{IIIC} corresponding to opening (I), sliding (II) and tearing (III) modes. The equivalent thermodynamic force Y takes the following form:

$$Y = \sup_{\tau \le t} G_{IC} \left\{ \left(\frac{Y_I}{G_{IC}} \right)^{\alpha} + \left(\frac{Y_{II}}{G_{IIC}} \right)^{\alpha} + \left(\frac{Y_{III}}{G_{IIIC}} \right)^{\alpha} \right\}^{1/\alpha} \qquad (2)$$

In the model, the three damage variables have the same evolution over the loading, and a unique damage d is therefore managed for modeling delamination, that is $d = d_I = d_{II} = d_{III}$. The damage variable d, considering the failure state at the interface between plies, is related to the equivalent thermodynamic force Y with a function of the form $g(Y)$. In SAMCEF, three different functions $g(Y)$ are available [14], leading to three possible cohesive laws, i.e. exponential, bi-triangular and polynomial. With this approach, it is possible to estimate the critical cracks, the propagation load, the maximum load the structure can sustain before a significant decrease of its strength and stiffness, and the residual stiffness during the inter-laminar cracks propagation.

In order to identify the values of the parameters entering the cohesive models in the interface, DCB, ENF and MMB tests are conducted. The corresponding finite element models are developed (Figure 6), and a fitting between experimental tests and numerical results is conducted, as explained in Figure 7. The analytical solutions based on the beam theory are also used to tune the parameters of the material models assigned to the interface.

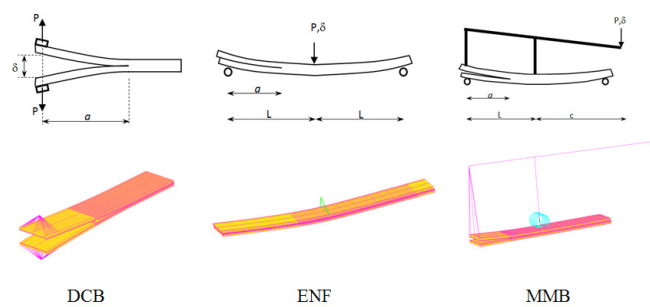

Figure 6. DCB, ENF and MMB tests and the corresponding finite element models

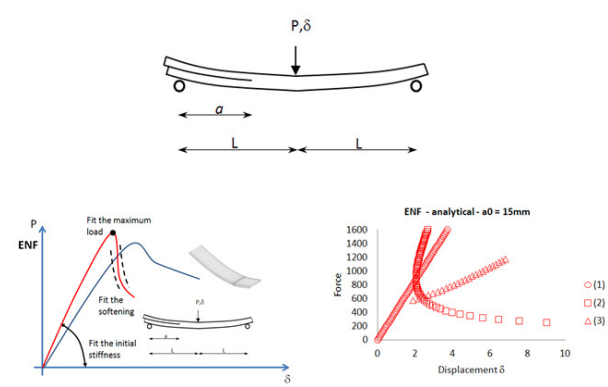

Figure 7. The principle of the parameter identification, explained on the ENF test case.

In this paper, the bi-triangular cohesive law is used. The different parameters that must be identified are the fracture toughness, the

initial stiffnesses and the interface strengths. Figures 8 to 10 represent the numerical responses when the interface parameters have been determined with the fitting process. Here, the coupling parameter of (2) is equal to 1, and the MMB test is used to validate the parameters values determined based on DCB and ENF.

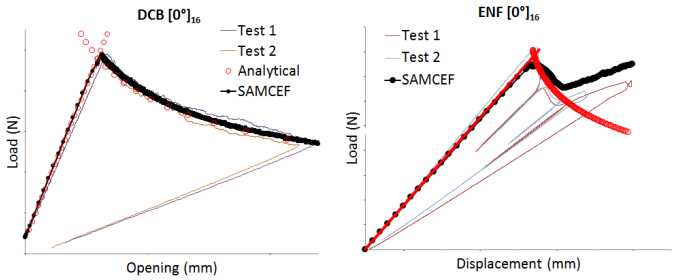

Figure 8. Results for DCB and ENF, for a $[0]_{16}$ stacking sequence [22,24]

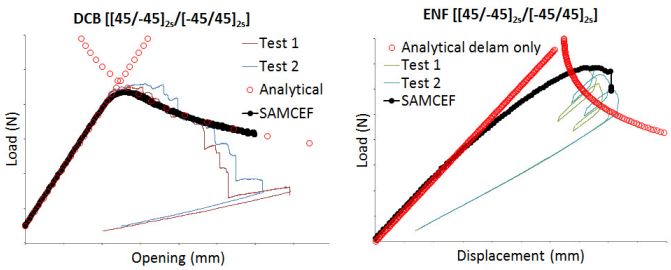

Figure 9. Results for DCB and ENF, for a stacking sequence including 45° and -45°layers [22,24]

It is observed in Figure 8 that for a $[0]_n$ laminate the behavior of the ENF test is quasi-linear up to the crack propagation load, which is the maximum point of the reaction-displacement curve. However, when the laminate includes ±45° orientations, the non-linear behavior observed in the tests can only be reproduced when the damage inside the plies is modeled. Doing so, we note a very good agreement between tests (light lines) and simulation (dark spots). For DCB, intra-laminar damage is not observed. The next section describes the strategy for the progressive damage modeling inside the plies.

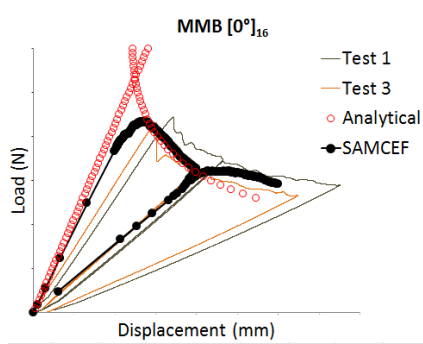

Figure 10. Validation on the MMB [22,24]

Modeling intra-laminar damage

Although delamination is certainly the most frequent mode of failure in laminated composites, in practical applications it is necessary to consider the ply degradation as well, as just demonstrated from Figure 9. Besides the classical failure criteria such as Tsai-Hill, Tsai-Wu and Hashin, an advanced degradation model is available in SAMCEF. This ply damage model relies on the development proposed in Ladeveze and LeDantec [7]. For intra-laminar damage, the following potential with damage, named e_d, is used (3), where d_{11}, d_{22} and d_{12} are the damages related to the fibers, the transverse and the shear directions, respectively (Figure 11).

$$e_d = \frac{\sigma_{11}^2}{2(1-d_{11})E_1^0} - \frac{\nu_{12}^0}{E_1^0}\sigma_{11}\sigma_{22} - \frac{\langle\sigma_{22}\rangle_+^2}{2(1-d_{22})E_2^0} + \frac{\langle\sigma_{22}\rangle_-^2}{2E_2^0} + \frac{\sigma_{12}^2}{2(1-d_{12})G_{12}^0} \quad (3)$$

The thermodynamic forces are derived from this potential and manage the evolution of the damages, via relations such as $d_{11} = g_{11}(Y_{11})$, $d_{22} = g_{22}(Y_{12},Y_{22})$ and $d_{12} = g_{12}(Y_{12},Y_{22})$. A delay effect can also be defined in order to smooth the occurrence of the damages. Moreover, non-linearities are introduced in the fiber direction, in tension and compression. For instance, the thermodynamic force associated to shear is given by:

$$Y_{12} = -\frac{\partial e_d}{\partial d_{12}} = \frac{\sigma_{12}^2}{2(1-d_{12})^2 G_{12}^0}$$

In Figure 12, it is seen that for a laminate submitted to pure shear (σ_{12},γ_{12}), a decrease in the stiffness is observed after some loading/unloading scenarios of increased amplitude, reflecting that damage occurs in the matrix. Moreover, unloading reveals the existence of permanent deformation, which is taken into account via a plasticity model. On top of that, non-linearities are introduced in the fiber direction, in tension and compression (Figure 13). It is noted from equation (3) that in the transverse direction, only tension leads to damage, but not compression, assuming the unilateral action of damage in direction 2 (crack closure in the matrix in compression). These behaviors result from the tests interpretation [7].

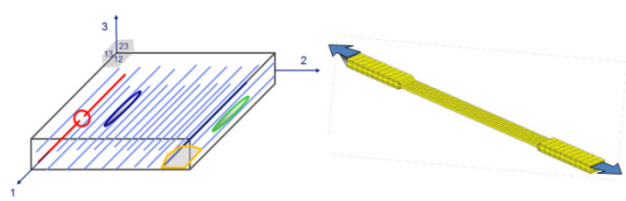

Figure 11. Possible damages in a UD ply, impacting fiber failure, matrix cracking and de-cohesion between fibers and matrix; model of the coupon.

From the coupon testing conducted on standard machines according to some standards like ASTM and equipped with strain gauges, the longitudinal stress σ_L and the axial and transversal strains (ε_L and ε_T) are obtained. Based on this information, the material behavior in each ply can be determined. Four series of tests are conducted, each one on a specific stacking sequence and/or loading scenario. As 5 successful tests are usually required, it means that 20 successful tests must be conducted to cover the 4 series. This is enough to identify the parameters of the progressive damage ply model as well as the elastic properties. The identification procedure is done without extensive use of simulation. It is rather a procedure based on EXCEL sheets, which can be speed up by using some very simple FORTRAN programming. A comparison between tests and simulation is used to validate the identified values.

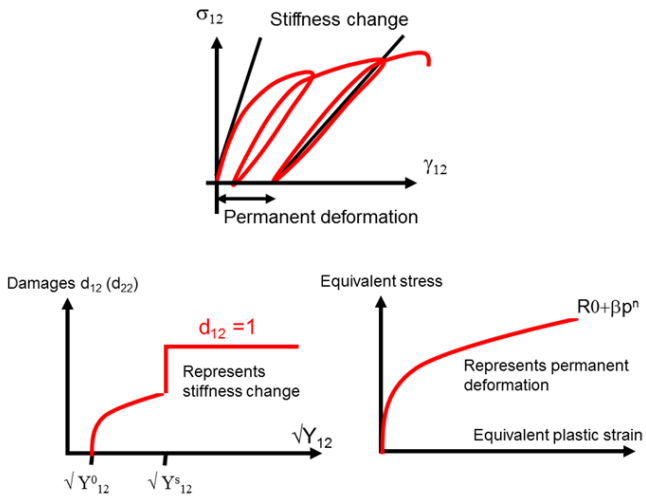

Figure 12. Non-linear behaviors captured by the SAMCEF intra-laminar damage model; matrix behavior

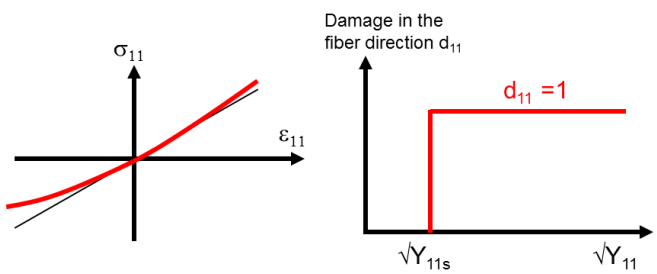

Figure 13. Non-linear behaviors captured by the SAMCEF intra-laminar damage model; fiber direction

The required stacking sequences mentioned above are not arbitrary; they are instead well defined, in order to be able to identify the whole set of elastic properties, the evolution of the damage and of the non-linearities of the material. One of these specific stacking sequences is made of plies at ±45°. The loading scenario is either classical, meaning that the coupon is loaded up to the final failure, or it is based on the loading/unloading (cyclic) sequences as described in Figure 12. As an example, a $[\pm 45]_{2s}$ laminate is studied. Based on the tests results as given in Figure 14, the evolution of the damage variable d_{12} is plotted as a function of the equivalent thermodynamic force Y_{12}. The hardening law of the plastic model is also identified. This allows determining the curves of Figure 12, which then feed the material model of SAMCEF. In Figure 15, it is checked that the results obtained with SAMCEF are in a very good agreement with the test results, not only for the global non-linear behavior, but also for the failure load estimation, the damage evolution (stiffness decrease measured during unloading) and the permanent deformation (plasticity). It is clear from Figure 15 that plasticity can't be neglected when studying polymer matrix composites. The non-linearities (hysteresis) appearing during loading/unloading, which is certainly due to friction between fibers and matrix, are not taken into account in the model. Our experience is that it doesn't influence the results. A specific angle ply laminate is considered to identify the material behavior in the transverse direction, which is actually coupled to shear. In order to take into account the coupling between shear and transverse effects, an equivalent thermodynamic force Y is used (4), and the evolution of d_{22} is, moreover, proportional to d_{12}:

$$Y = Y_{12} + bY_{22} \quad \text{and} \quad d_{22} = cd_{12} \tag{4}$$

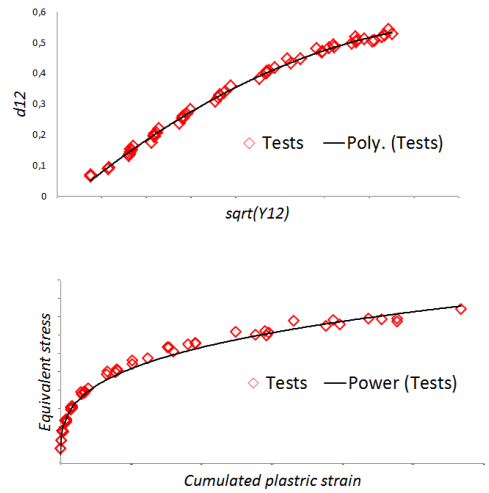

Figure 14. Identification of the damage evolution in shear and of the hardening for the plastic behavior [23,24].

The material behavior in the pure transverse direction is also identified, as illustrated in Figure 16 (left). The resulting damage laws evolutions are also given in Figure 16 (right). This information feeds the progressive damage ply model of SAMCEF.

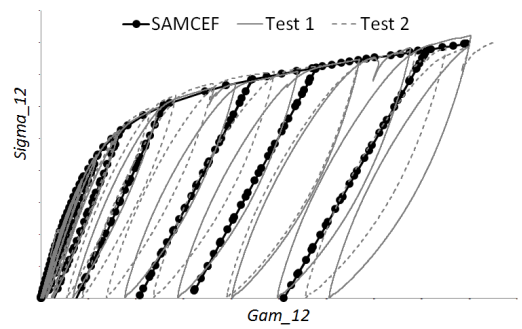

Figure 15. Comparison between tests and simulation results for a laminate made up of ±45° plies [23,24]

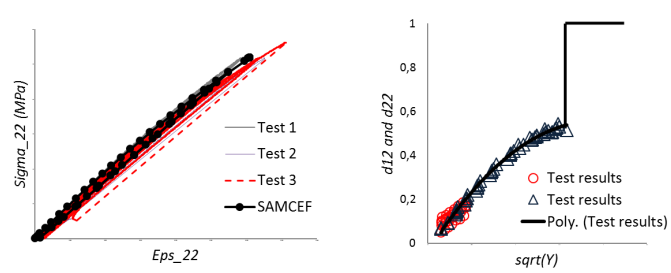

Figure 16. Transverse behavior and full intra-laminar damage evolution [23,24].

In Figure 17, the evolution of the stiffness modulus E_{11} in the fiber direction is identified, in tension (hardening effect) and compression (softening effect). The softening effect appearing in compression is (partly) due to fiber micro-buckling. The (very small) hardening

effect in traction is related to the alignment of the fibers in the loading direction. The failure loads in the fiber direction are also easily determined based on the tests, in tension and in compression. In Figure 18, the longitudinal force F_L and the corresponding longitudinal strain in the coupon are plotted. This allows determining the strength in the fiber direction. Here, the force is applied on the coupon in the model, and the displacement becomes very large when the maximum load has been reached (force controlled analysis).

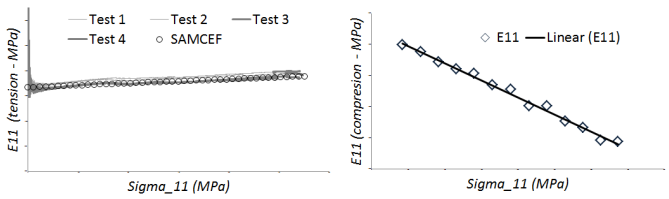

Figure 17. Identification of the E_{11} coefficient non-linear behavior [23,24].

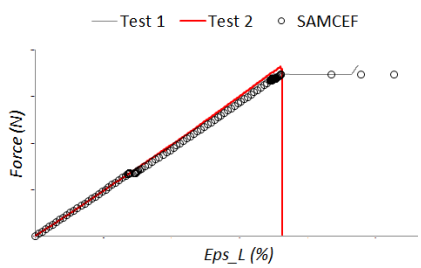

Figure 18. Identification of the behavior in the fiber direction [23,24].

From Figure 19, it is clear that when inaccurate values of the parameters are used in the progressive ply damage model, simulation results are not in a good agreement at all with the tests. In Figure 19, the damage and plasticity laws of Figure 14 were modified, as well as the strength. This solution should be compared to the one obtained in Figure 15. Comparing those two Figures, it is clear that an accurate identification procedure is necessary if one wants to be able to reproduce the non- linear behavior of the composite material.

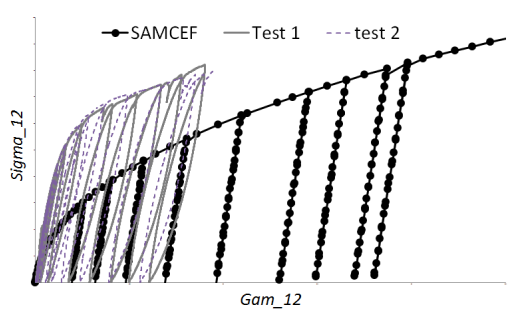

Figure 19. Numerical simulation and comparison to tests for wrong values of the parameters.

In order to validate the value of the parameters of the progressive damage model, a blind test is conducted on a $[67.5/22.5]_{2s}$ coupon. This stacking sequence was not used for the parameter identification. Simulation is run, and a comparison to test results is done. A very good agreement is obtained, as illustrated in Figure 20. Compared to the initially identified value of the parameters, just the failure load in the transverse direction had to be a little bit increased. The values of the progressive ply damage model parameters are then validated, and can therefore be used to study any coupon made of an arbitrary number of plies and arbitrary orientations. The only restrictions are that the base material properties (of the fibers and the matrix) and the fiber volume fraction can't be changed, and that the properties are obtained for given temperature and humidity levels. This information can now be used to predict the behavior of more general composite parts on the upper stages of the pyramid (Figures 1 and 2).

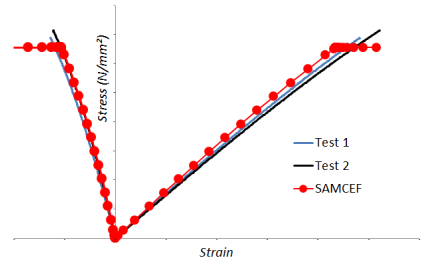

Figure 20. Validation of the parameter identification process [23,24].

Up in the pyramid of tests: impacted plate

As illustrated in Figure 3, when a laminated composite plate is submitted to a low energy impact, damage will appear, mainly at the interfaces between the plies, although it is not visible from the outside. This situation is of course very dangerous, and must be taken into account during the sizing in order to determine allowables depending on the damage tolerance capacity of the composite.

Here, a specimen of 100x100mm² is mounted on the impact test device. The plate is clamped at its 4 edges. The impactor has a diameter of 16mm and is made of metal. The impact energy is a function of the total thickness and corresponds to 3.34J/mm. We are here in the case of an invisible damage resulting from an impact (Figure 3, left). The problem is solved as a quasi-static load case. This assumption has been validated by a comparison with the results obtained with SAMCEF and its dynamic analysis capabilities: the amount of kinetic energy was negligible in the problem. The composite plate is built with the following stacking sequence: $[45/0/-45/90]_s$. One solid finite element is used on the ply thickness, so 8 elements are used on the laminate thickness to represent the plies and 7 interfaces are defined.

A comparison of the inter-laminar damage obtained from tests and simulation is given in Figure 21. The test results are obtained with a C-scan. This method is used to identify the defects appearing in the laminate. It was actually unable to determine in which interface the defect was located, but rather provided the successive identified damages through the thickness. In the simulation results, red means completely broken, while blue means no damage of the interface.

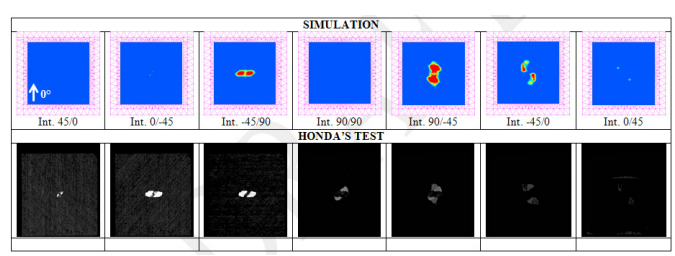

Figure 21. Test and numerical results for the impact on a laminated plate made of 8 plies

In Figure 22, a correspondence is done between the defects determined with the C-scan and the numerical results obtained with SAMCEF. A very good agreement is observed.

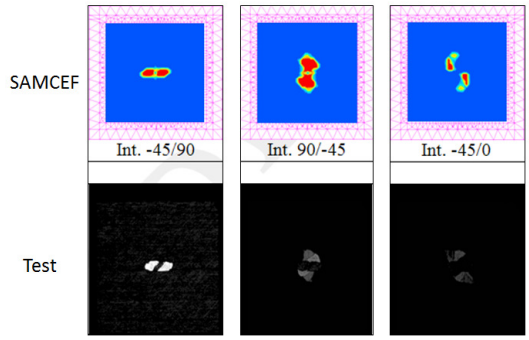

Figure 22. Test and numerical results for the impact on a laminated plate made of 8 plies; correspondence between C-scan and simulation.

A $[45/0/-45/90]_{3s}$ laminate was also studied, as illustrated in Figure 23. Even if some similarities are clearly observed, it is anyway more complicated to make a direct link between test and simulation results because of the large number of interfaces. Only 9 pictures are taken on each side of the laminate (named "front" for the top face and "back" for the bottom face).

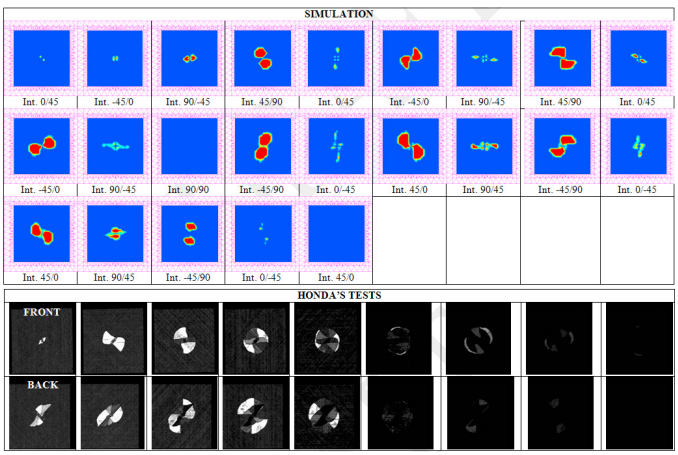

Figure 23. Test and numerical results for the impact on a laminated plate made of 24 plies.

Based on these good results, SAMCEF was then used as a predictive simulation tool to estimate the damage appearing in laminates made of different stacking sequences, for which no physical tests were conducted.

Up in the pyramid of tests: L-shaped beam

The L-shaped beam is submitted to an imposed vertical displacement on its upper face and is clamped on its vertical leg, as illustrated in Figures 24 and 25. The laminated L-shaped beam is made of 12 plies, with the following stacking sequence $[60/-60/0/0/-60/60]_s$. The developed model is illustrated in Figure 25. Contact elements are used and rigid bodies transmit the loading to the composite part. One solid finite element is used on the ply thickness, so 12 elements are used on the laminate thickness to represent the plies and 11 interfaces are defined.

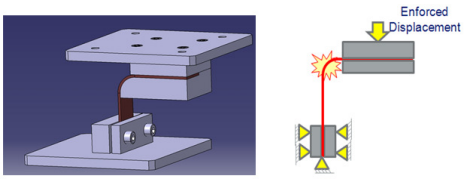

Figure 24. The L-shaped beam and the boundary conditions.

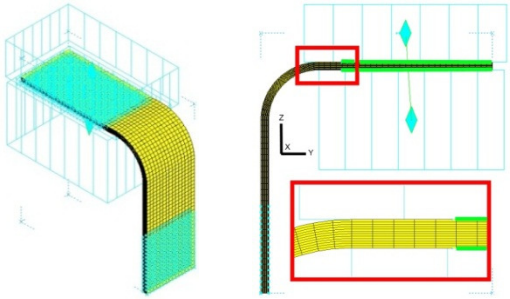

Figure 25. The model of the L-shaped beam.

Deformed configurations are illustrated in Figure 26. The global deformations obtained with the numerical method are in good agreement with the pictures taken during the physical test. Anyway, as for the available pictures the loading amplitude associated to the test results is not known, only a quantitative comparison can be made. From simulation, it is observed that even if damage appears inside the plies delamination is predominant in this case. The load-displacement curve is given in Figure 27, where a comparison is done between test and simulation results.

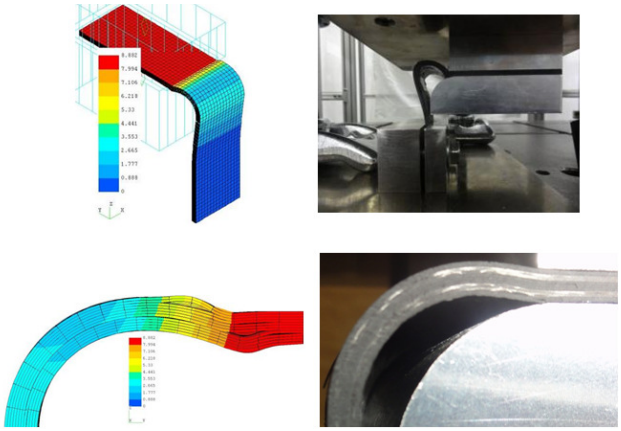

Figure 26. The L-shaped beam in two loading configurations

Conclusions

In this paper, the non-linear behavior of laminated composite materials and structures made of UD plies was studied. Physical tests and numerical analyses were conducted. The specific damage models available in the SAMCEF finite element code were used for modeling the inter- and intra-laminar damages.

First, tests results at the coupon level are used to identify the value of the parameters of the damage models. These values are validated at the coupon level. Then, the material models are used to study more complicated composite structures and/or loading at upper stages of

the pyramid of tests. The cases of an impacted plate and of a L-shaped beam submitted to bending are studied. The very good agreement obtained between simulations and tests tend to demonstrate that SAMCEF can be used as a predictive numerical tool for the evaluation of the non-linear behavior of composites including damage.

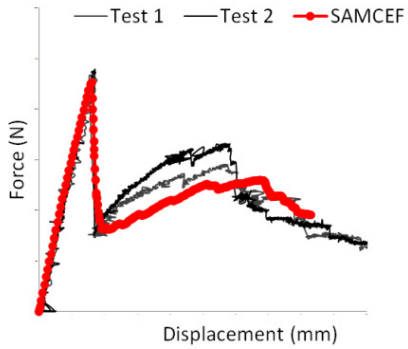

Figure 27. The L-shaped beam load-displacement curve: tests and simulation.

References

1. Katerelos, D.D.G., Kashtalyan, M., Soutis, C., Galiotis, C. "Matrix cracking in polymeric composites laminates: modeling and experiments". *Composite Science & technology*, 68: 2310-2317, 2008.
2. Tan, S.C. and Nuismer R.J. "A theory for progressive matrix cracking in composite laminates", *J. Compos. Mater.*, 23: 1029–47, 1989.
3. Li S., Reid S.R. and Soden P.D. "A continuum damage model for transverse matrix cracking in laminated fibre-reinforced composites", Philos Trans R Soc Lond Ser A (Math Phys Eng Sci), 356: 2379–412, 1998.
4. Adolfsson, E. and Gudmundson, P. "Matrix crack initiation and progression in composite laminates subjected to bending and extension", *Int. J. Solids Struct.*, 36: 3131–69, 1999.
5. Katerelos, D.T.G, Varna, J. and Galiotis, C. "Energy criterion for modeling damage evolution in cross-ply composite laminates", *Compos Sci Technol*, 68: 2318–24, 2008.
6. Mayugo, J.A., Camanho, P.P., Maimi, P. and Dávila, C.G. "Analytical modeling of transverse matrix cracking of [±h/90n] composite laminates under multiaxial loading", *Mech Adv Mater Struct*, 17: 237–45, 2010.
7. Ladeveze, P. and Le Dantec, S. "Damage modeling of the elementary ply for laminated composites", *Composites Science & Technology*, 43: 123-134, 1992.
8. Krüeger, R. 2007. "An approach for assessing delamination propagation capabilities in commercial finite element codes", American Society of Composites 22nd Annual Technical Conference, University of Washington, Seattle, WA, September 17-19, 2007.
9. Camanho, P.P. 2006. "Finite element modeling of fracture in composites: current status and future developments", NAFEMS Seminar – Prediction and Modeling of Failure Using FEA. NAFEMS, Roskilde, Denmark, 2006.
10. Krüeger, R. "Virtual Crack Closure technique: history, approach and applications". *Applied Mechanics Reviews* 57: 109-143, 2004.
11. Krüeger, R. "Three dimensional finite element analysis of multidirectional composite DCB, SLB and ENF specimens", *ISD-Report N° 94/2*, Institute for Statics and Dynamics of Aerospace Structures, University of Stuttgart, 1994.
12. Orifici, A.C., Thomson, R.S., Degenhardt, R., Bisagni, C., and Bayandor, J. "Development of a finite element analysis methodology for the propagation of delaminations in composite structures". *Mechanics of Composite Materials*, 43: 9-28, 2007.
13. Xie, D. and Bigger, S.B. "Progressive crack growth analysis using interface element based on the Virtual Crack Closure Technique". *Finite Elements in Analysis and Design* 42: 977-984, 2006.
14. Bruyneel, M., Delsemme, J.P., Jetteur, P. and Germain, F. "Modeling inter-laminar failure in composite structures: illustration on an industrial case study", Applied Composite Materials, 16: 149-162, 2009.
15. Allix, O. and Ladevèze, P. "Interlaminar interface modelling for the prediction of laminate delamination", Composite Structures, 22: 235-242, 1992.
16. Bruyneel M. and Diaconu C. "Structural composite design: concepts and considerations", Encyclopedia of Composites, Wiley & Sons, 2012.
17. Baker A., Dutton S., and Kelly D. "Composite Materials for Aircraft Structures", AIAA Education Series, Reston, 2004.
18. Farkas L., Liefooghe C., Bruyneel M., Hack M., Van Genechten B., Donders S., and Van der Auweraer H. "Roadmap for addressing the automotive engineering challenges in composite development", Tools and Methods of Competitive Engineering (TMCE2014), Budapest, Hungary, May, 2014.
19. Bruyneel M., Delsemme J.P., Jetteur P., Germain F. and Boudjemaa N. (2014). "Damage modelling of composites: validation of inter-laminar damage model at the element level", JEC Composites Magazine, 90, June 2014.
20. Bruyneel M., Degenhardt R. and Delsemme J.P. "An industrial solution to simulate post-buckling and damage of composite panels", JEC Composites Magazine, 48, May 2009.
21. Galucio A.C., Jetteur P., Trallero D., Charles J.P. "Toward numerical fatigue prediction of composite structures : application to helicopter rotor blades", 3rd ECCOMAS Conference on the Mechanical Response of Composites, 21-23, Hannover, Germany, 2011.
22. Bruyneel M., Delsemme J.P., Goupil A.C., Jetteur P., Lequesne C., Naito T., Urushiyama Y. "Damage modeling of laminated composites : validation of the inter-laminar damage law of SAMCEF at the coupon level for UD plies", World Congress of Computational Mechanics, WCCM11, Barcelona, Spain, 20-25 July, 2014.
23. Bruyneel M., Bruyneel M., Delsemme J.P., Goupil A.C., Jetteur P., Lequesne C., Naito T., Urushiyama Y. "Damage modeling of laminated composites : validation of the intra-laminar damage law of SAMCEF at the coupon level for UD plies", European Conference on Composite Material, ECCM16, Sevilla, Spain, 22-26 June, 2014.
24. Bruyneel M., Delsemme J ;P., Goupil A.C., Jetteur P., Lequesne C., Naito T., Urushiyama Y. "Validation of material models for inter and intra-laminar damage in laminated composites", 2014 ASC 29/US-Japan 16 ASTM D30 Conference, UC, San Diego, USA, September 2014.

Contact Information

In case of question or need for additional information on this work, please contact Dr. Yuta Urushiyama (Honda R&D, Tochigi, Japan: Yuta_Urushiyama@n.t.rd.honda.co.jp) and Prof. Dr. Michael Bruyneel (SAMTECH, A Siemens Company, Liège, Belgium: Michael.bruyneel@siemens.com).