11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

NONLINEAR PREDICTIONS IN LAMINATED COMPOSITES AND STRUCTURES

HEINZ E. PETTERMANN*, JOHANNES BROGER*,† AND JAKOB GAGER*,†

* Institute of Lightweight Design and Structural Biomechanics,
Vienna University of Technology, A-1040 Vienna, Austria,
e-mail: pettermann@ilsb.tuwien.ac.at, web page: http://www.ilsb.tuwien.ac.at

† Polymer Competence Center Leoben GmbH,
A-8700 Leoben, Austria,
web page: http://www.pccl.at

Key words: Continuous Fiber Reinforced Polymers, Continuum Damage Mechanics, Plasticity, Constitutive Material Model, Structural Analysis, Finite Element Method

Abstract. The nonlinear mechanical response of laminated composite structrures is investigated by the Finite Element Method. Each ply is modeled by its own layer of shell elements and these layers are connected by cohesive elements. The latter allow for damage and failure of the interface between the plies. The constitutive behavior of the unidirectional reinforced plies is modeled by an in-house material law developed earlier. It accounts for progressive damage and failure as well as plasticity effects and has been implemented for implicit solution schemes. The model is converted to suit explicit dynamic Finite Element Methods and is applied to an Open Hole Tension test scenario. Quasi-static predictions are carried out for a quasi-isotropic laminat, for which both progressive ply and progressive interface damage is considered simultaneously.

1 INTRODUCTION

Continuous fiber reinforced polymer laminates have found widespread application in high tech products. Their high specific stiffness and strength make them excellent choices for lightweight components. Challenges may arise from the complex behavior of the plies beyond the elastic limit and the wide variety of nonlinear mechanisms as well as the pronounced directional sensitivity. To better exploit the potential of such composites, appropriate constitutive material models are required to predict their response. Moreover, such models need to be suited for utilization in structural analyses to compute the behavior of laminated composite components. This should exceed classical "first ply failure" considerations to evaluate stress redistribution and, consequently, component's strength

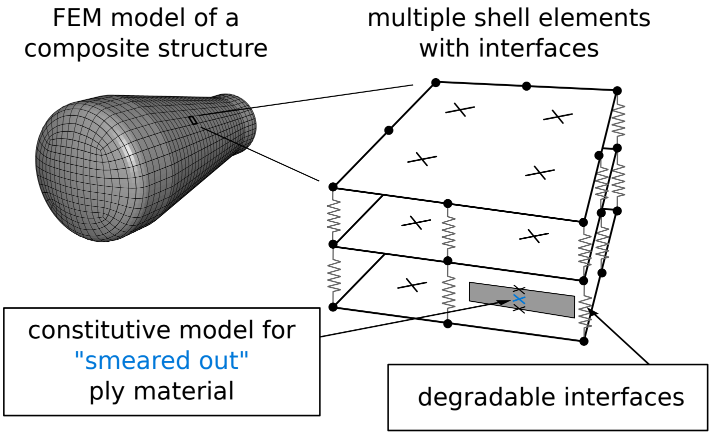

Figure 1: Shell modeling concept accounting for intra and inter ply nonlinearities.

reserve and peak load. Additionally, it may be of interest whether damage growth in the component is stable or unstable.

In the present paper a nonlinear constitutive material law is employed to model the plies' behavior. For the simulation of delamination between the plies, a cohesive contact formulation is used. Both simulation tools are combined within the framework of the Finite Element Method which allows for the structural analyses of components accounting for a variety of sources of nonlinearity simultaneously. The approach has been applied to a number of problems; here, the nonlinear response of the Open Hole Tension test is predicted.

2 MODELING APPROACH

The general approach to modeling applied here has been introduced in [1], and is adopted for the present study. This approach consists of the usage of shell elements to model the individual plies and cohesive contact elements to model the interfaces in between, Fig. 1. This way the constitutive behavior of both the plies and their interfaces can be modeled by a variety of nonlinear models. The underlying approximation is that the tri-axial stress state is decomposed in the following sense. The in-plane stress components act in the shell elements, whereas the complementary components state the interface tractions. For these two types many constitutive models are available within commercial Finite Element programs.
The simulations are carried out with ABAQUS/Explicit 6.13 (*Dassault Systèmes Simulia Corp., Providence, RI, USA*). Delamination is modeled by ABAQUS on-board means.
For the plies a nonlinear constitutive material law [2] is employed which models the mechanical response of a unidirectional reinforced composite ply under plane stress assumption. It is based on a phenomenological description in terms of continuum damage mechanics enriched by plasticity mechanisms. An older version of this model also participated in the "Third World Wide Failure Exercise" [3]. The constitutive model has

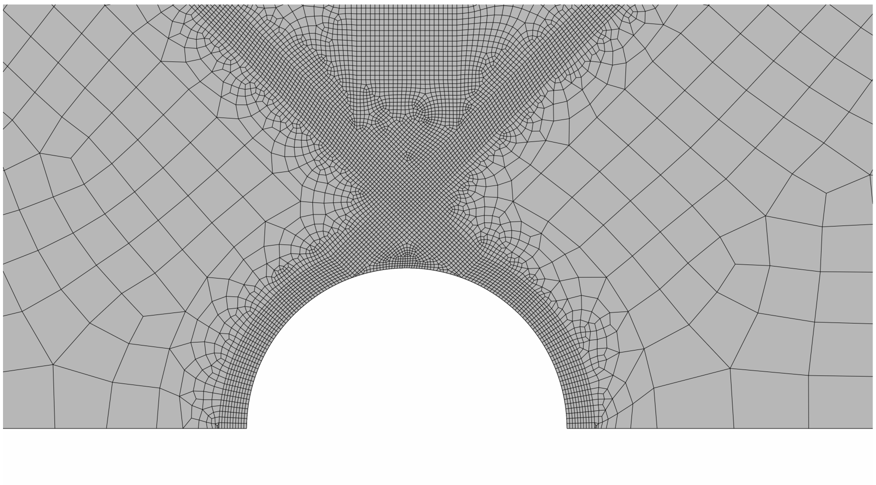

Figure 2: Detail of the Finite Element mesh around the hole.

been implemented for implicit integration schemes within the Finite Element Method, i.e. in ABAQUS/Standard. For the present study it is converted to be suited for explicit dynamic simulations, for details see [4]. This enlarges the applicability of the constitutive model, e.g. to impact problems. The present investigations, however, concern quasi-static problems solved by ABAQUS/Explicit.

Various applications of this modeling approach are shown spanning from predictions for unidirectional reinforced layers to structural analyses. The latter comprise Open Hole Tension Tests [5] and L-shaped laminates in which ply nonlinearities as well as delamination mechanisms are considered simultaneously [6]. Also, the approach is employed in the simulation of textile composites modeling yarns in woven or braided layers [1].

3 EXAMPLE — OPEN HOLE TENSION TEST

As example the Open Hole Tension test is modeled corresponding to the simulations in [5]. A quasi-isotropic lay-up of $[0/{-45}/90/{+45}]_S$ is considered, the width is 32.5 mm, the length between the tabs is 130 mm, and the central hole has a diameter of 6.4 mm. The specimen is loaded in uniaxial tension.

3.1 Finite Element Model

In the Finite Element model only the central part with a length of 50 mm is modeled. Symmetries in the width direction as well as in the thickness direction is applied. It is noted that symmetries have to be used with care when damage and failure is considered.

The mesh is the same in all layers, a zoom-in at the hole is shown in Fig. 2. At the vicinity of the hole a typical element length of 0.05 mm is used. The same size is used in regions where localization is expected. However, no measures whatsoever need to be taken to start or initialize damage.

Four-noded shell elements are used, they are reduced integrated with five section points

Table 1: Essential material data for the unidirectional carbon-epoxy ply and the interface.

UD carbon-epoxy ply				
E_1	$E_2 = E_3$	$\nu_{12} = \nu_{13}$	ν_{23}	$G_{12} = G_{13}$
146 GPa	9 GPa	0.34	0.61	4.27 GPa
R^{t}_{11}	R^{c}_{11}	R^{t}_{22}	R^{c}_{22}	R_{12}
2100 MPa	1407 MPa	82 MPa	249 MPa	110 MPa
$G_{(ft)}$	$G_{(tc)}$	$G_{(mt)}$	$G_{(mc)}$	$G_{(ps)}$
89800 J/m^2	78300 J/m^2	200 J/m^2	800 J/m^2	1000 J/m^2
Viscous regularization parameters				
$\eta^{\mathrm{ft}}_{\mathrm{mat}} = \eta^{\mathrm{fc}}_{\mathrm{mat}}$	$\eta^{\mathrm{mt}}_{\mathrm{mat}} = \eta^{\mathrm{mc}}_{\mathrm{mat}}$			
0.002 s	0.004 s			
Interface				
t_n	t_s	G_I	$G_{II} = G_{III}$	
60 MPa	110 MPa	133 J/m^2	459 J/m^2	

in the thickness direction applying the Simpson rule integration. The material under consideration is the carbon/epoxy systems Cycom977, Tab. 1. Even though, viscous regularization is not necessary for explicit simulations it is used for better comparability to the results from the implicit solutions. The shell elements in use require the manual definition of the transverse shear stiffness, which is calculated according to the ABAQUS manual.

The interfaces are modeled with cohesive elements. Their constitutive behavior is described by a traction-separation response with the possibility to exhibit damage and failure. The initiation of damage is described by a quadratic stress criterion. Its evolution is modeled via the energy dissipated by the damage process, the mode mix is described by the Benzeggagh-Kenane criterion with an exponent of 2. The properties of the cohesive interface are listed in Tab. 1. The cohesive elements and the shell elements are coupled together via tie constraints. Elements are removed once the maximum damage is reached.

Residual stresses from curing are introduced by modeling a homogeneous temperature reduction of 157 K from the stress free state. The tensile load is applied as uniform displacements at both sides with equal magnitude in load direction.

3.2 Results

The global load displacement behavior is nearly linear, although the simulation shows highly nonlinear behavior of the plies and interfaces locally. The first occurrence of interface degradation is predicted at a load of only 2.5 kN, restricted to a very small area at the vicinity of the hole. At a load level of 4.5 kN the damage of the interface has not grown significantly, but first signs of plasticity are observed in the 0°, +45°, and –45° ply.

Distributed transverse damage accumulation starts at a load of 10.8 kN in the 45° and 90° ply.
At a load level of 17.0 kN the +45° and the –45° ply are almost completely affected by plasticity. Transverse damage localization is exhibited in the –45° ply, onset of fiber damage is observable in the 0° ply. Some occurrence of delamination is exhibited at the hole but limited to very small areas in the vicinity of the free edge.
Load increase leads to damage localization first in the –45° ply, followed by the 90°, the +45°, and the 0° ply.
At the structural peak load of 32.0 kN the pattern of transverse ply damage is shown in Fig. 3. The 0° ply exhibits combined transverse and fiber damage in a zone perpendicular to the loading direction. This developing damage zone limits the load carrying capacity of this specimen. But at this load level the influence of viscous regularization is already substantial, i.e. the volume specific energy dissipated through viscous regularization is about half the value of the volume specific energy dissipated by damage in this localization areas. Therefore, the peak load might be overestimated. The +45° ply behaves as expected, a localization zone in fiber direction evolves. The interfaces between the plies are damaged in wide areas at this load level, see Fig. 4. Due to this delamination areas, the damage behavior of the individual plies is not coupled any more allowing for individually evolving localization patterns. In the vicinity of the hole, the out-of-plane displacement of the 90° and + 45° plies can be explained by these delaminated areas.

4 CONCLUSIONS

The Finite Element Method is employed to predict the nonlinear mechanical response of an Open Hole Tension test as example for a structural problem. A quasi-isotropic laminate is investigated for which both progressive ply and interface damage is considered simultaneously.
A shell modeling approach is adopted by which each ply is modeled by its own layer shell elements. These layers are connected by cohesive elements which allow for interface damage and failure. The constitutive behavior of the unidirectional reinforced plies is modeled by an in-house material law. It accounts for progressive damage and failure as well as plasticity effects. For the present study the model is converted to suit explicit dynamic Finite Element Methods.
Quasi-static predictions are carried out for a structural problem. The analyses successfully show the combination of ply damage and delamination as well as their interaction. The sequence of occurrence of various nonlinear effects can be predicted, including the triggering of some damage modes by preceding nonlinear effect caused by some other damage or failure mode. This way, not only the nonlinear behavior of the structure is simulated and the local load redistribution, but also the peak load and post-peak response can be predicted.

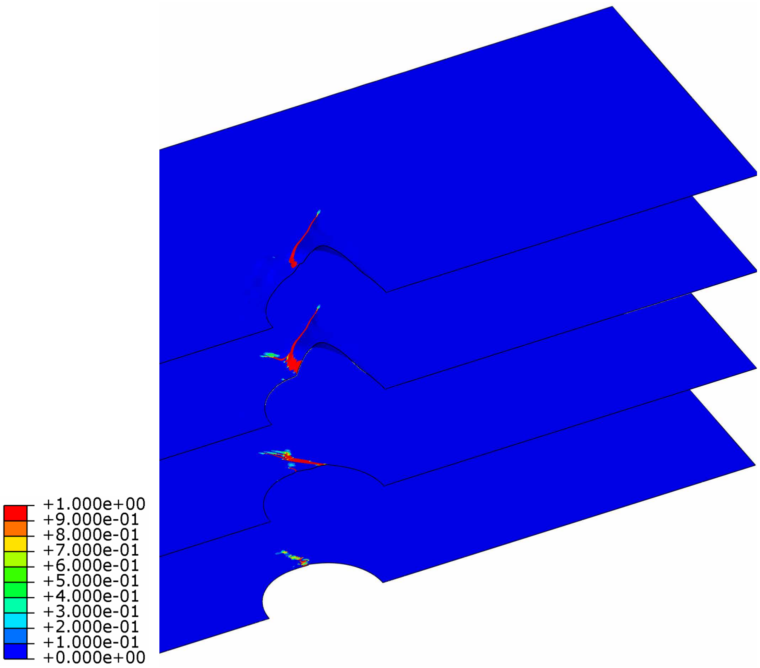

Figure 3: Predicted localization zones in terms of the transverse damage variable in the plies at peak load (F=32.0 kN); displayed plies from bottom to top: 0/−45/90/+45, failed elements not shown.

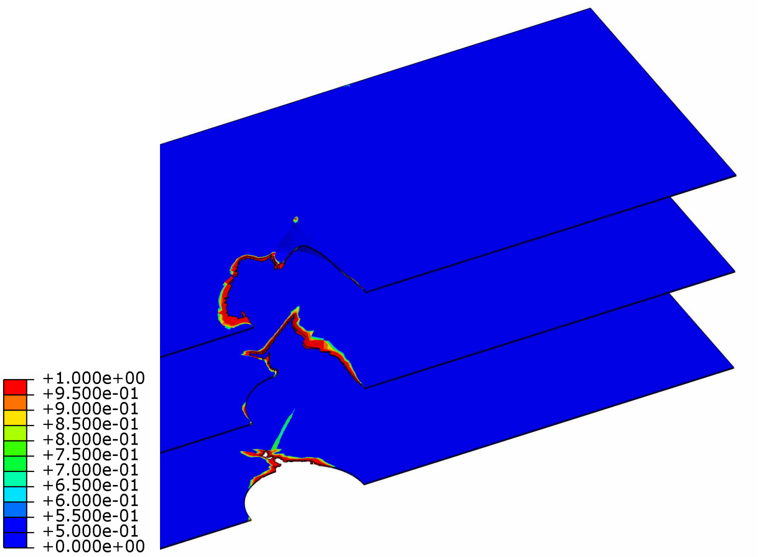

Figure 4: Predicted delamination zones in terms of the interface damage variable at peak load (F=32.0 kN); displayed interfaces from bottom to top: 0/−45, −45/90, 90/+45, failed elements not shown.

Acknowledgment

The funding of the Polymer Competence Center Leoben GmbH (PCCL, Austria) within the framework of the COMET-K1-program of the Austrian Ministry of Traffic, Innovation, and Technology and the Austrian Ministry of Economics and Labor is gratefully acknowledged.

REFERENCES

[1] Gager J., Pettermann H.E. Numerical homogenization of textile composites based on shell element discretization. *Compos.Sci.Technol.*, **72**, 806–812, 2012.

[2] Flatscher Th., Pettermann, H.E. A constitutive model for fiber-reinforced polymer plies accounting for plasticity and brittle damage including softening - implementation for implicit FEM. *Compos.Struct.*, **93**, 2241–2249, 2011.

[3] Flatscher Th., Schuecker, C., Pettermann, H.E. A constitutive ply model predicting stiffness degradation as well as inelastic strain accumulation and its application to WWFE-III (Part A). *J.Comp.Mater.*, **47**, 2575-2593, 2013.

[4] Broger J. *Adaptation of an Elasto-Plasto-Damage Model from Abaqus/Standard to Abaqus/Explicit and its Application to Laminated FRP Composites.* Master Thesis, Vienna University of Technology, Vienna, Austria, 2014.

[5] Flatscher Th., Wolfahrt M., Pinter G., Pettermann H.E. Simulations and Experiments of Open Hole Tension Tests Assessment of Intraply Plasticity, Damage, and Localization, *Compos.Sci.Technol.* **72**, 1090–1095, 2011.

[6] Pettermann H.E., Gager J., Rama E., Flatscher Th. Elasto-Plasto-Damage Modeling of Laminated Composites. In *Proceedings of the* 6^{th} *European Congress on Computational Methods in Applied Sciences and Engineering*, J. Eberhardsteiner et.al. (eds.) Vienna, Austria, 2012.