Joris J.C. Remmers, Albert Turon
Editors

Proceedings of the sixth ECCOMAS Thematic Conference on the
Mechanical Response of Composites
Editors

Joris J.C. Remmers
Eindhoven University of Technology
Department of Mechanical Engineering
Mechanics of Materials
PO Box 513
5600 MB, Eindhoven
The Netherlands

Albert Turon
University of Girona
AMADE-Advanced Materials and Analysis
C/Universitat de Girona 4
17003, Girona
Spain

A catalogue record is available from the Eindhoven University of Technology Library
ISBN: 978-90-386-4366-3

Cover Design: Alice van Litsenburg

www.tue.nl/mechmat
www.tue.nl/composites2017
Preface

The 6th ECCOMAS Thematic Conference on the Mechanical Response of Composites, known as Composites2017, was held in Eindhoven, The Netherlands, on September 20-22, 2017. This three-day event attracted 120 delegates, from 20 countries, with 91 presentations over 19 sessions. This book contains the proceedings submitted by a number of authors.

Previous editions of this conference series were held in Porto (Portugal), London (UK), Hannover (Germany), Ponta Delgada (Portugal) and Bristol (UK) in 2007, 2009, 2011, 2013 and 2015 respectively. Over the years, this series of thematic conferences has become well-established in the Composites Materials community, with a significant number of participants having attended the majority of previous editions of this conference. The conference series covers the scientific investigation of complex mechanical behaviour of composite materials and structures. It focuses on theoretical and numerical modelling and prediction of the performance of composite components, also covering experimental validation and challenging industrial applications or recent developments.

In this sixth edition plenary lectures were given by Dr. Tong Earn Tay (National University of Singapore), Dr. Michel Fouinneteau (Airbus Operations SAS) and Dr. Pedro Camanho (University of Porto). The sessions at the event covered a wide range of topics, including Novel Numerical Techniques, Multi-Scale Modelling, Hybrid and Multi-Functional Composites, Textiles, Structures, Testing, Impact, Fatigue, Probabilistic methods, Design, Optimisation, Dynamic effects, Delamination and Matrix Cracking.

We would like to thank our local organising team, in particular Alice van Litsenburg, Jim Schormans and Rachel van Outvorst for their efforts in organising the conference and preparing these proceedings. We would also like to thank the European Community on Computational Methods in Applied Science and Engineering (ECCOMAS) for their support to the conferences and the Research Centre Fluid & Solid Mechanics of the Dutch 4TU federation for the financial support. Our gratitude goes to the members of the scientific committee for their help in reviewing the abstracts. Finally, we thank the keynote speakers and all the participants for their contributions and fruitful interactions during the conference. The next edition in this conference series will be hosted by the Universitat de Girona, Spain, in September 2019 (http://amade.udg.edu/composites2019).

Eindhoven, September 2017

Joris Remmers
Albert Turon

Conference chairs
Composites 2017
Organization

The sixth ECCOMAS Thematic Conference on the Mechanical Response of Composite Materials (Composites 2017) is organised by:

Department of Mechanical Engineering
Mechanics of Materials
Technische Universiteit Eindhoven

Conference chairs

Joris J.C. Remmers Technische Universiteit Eindhoven, The Netherlands
Albert Turon Universitat de Girona, Spain

Local Organisation

Alice van Litsenburg Technische Universiteit Eindhoven, The Netherlands
Jim M.J. Schormans Technische Universiteit Eindhoven, The Netherlands
Rachel van Outvorst Technische Universiteit Eindhoven, The Netherlands

Scientific Board

O. Allix ENS Cachan, France
C. Bisagni Technische Universiteit Delft, The Netherlands
R. de Borst University of Sheffield, UK
P.P. Camanho Universidade do Porto, Portugal
D. Coker METU, Turkey
A. Corigliano Politecnico di Milano, Italy
J. Costa Universitat de Girona, Spain
C.G. Dávila NASA Langley, USA
M. Fagerström Chalmers Tekniska Högskola, Sweden
Scientific Board (continued)

K. Gamstedt  Uppsala Universitet, Sweden
S.R. Hallett  University of Bristol, UK
L. Iannucci  Imperial College, UK
M. Kawai  University of Tsukuba, Japan
J. LLorca  IMDEA / UP Madrid, Spain
S. Lomov  KU Leuven, Belgium
E. Lund Aalborg  Universitet, Denmark
M. May  Fraunhofer Institute, Germany
C. McCartey  University of Limerick, Ireland
F. van der Meer  Technische Universität Delft, The Netherlands
P. Middendorf  Universität Stuttgart, Germany
A. Mouritz  RMIT University, Australia
E. Oñate  Universitat Politècnica de Catalunya, Spain
S. Pinho  Imperial College, UK
J.N. Reddy  Texas A&M, USA
R. Rolfes  Leibniz Universität Hannover, Germany
K. Schulte  TU Hamburg-Harburg, Germany
B.F. Sørensen  Danmarks Tekniske Universitet, Denmark
R. Talreja  Texas A&M, USA
R. Vaziri  University of British Columbia, Canada
A. Waas  University of Michigan, USA
M.R. Wisnom  University of Bristol, UK
Q.D. Yang  University of Miami, USA
VI

Sponsors

This conference is sponsored by the Research Centre Fluid & Solid Mechanics of the Dutch 4TU federation.
# Table of Contents

Analytical Axial Harmonic Response of a Composite Bonded Lap Joint ............................................. 1  
*Georges E. Challita, Ramzi A. Othman*

DCB and ENF Multi-Scale Simulations ............................................................................................... 13  
*Tilen Ceglar, Martin Schwab, Heinz E. Pettermann*

Interlaced Composites winding (D-3F technology) as an alternative for braiding and winding ...... 24  
*Daniel Denninger, Maik Berger, Yordan Kyosev*

High speed modelling of polyamid-matrix woven composites ....................................................... 36  
*Nicolas Feld, Fabien Coussa, Benoit Delattre, Jacques Renard*

A cyclic cohesive zone model for variable amplitude loading and mixed-mode behavior ............. 48  
*Martin Springer, Albert Turon, Heinz E. Pettermann*

Experimental investigation of a novel hybrid interlocking aluminium/composite joint at the single feature scale ................................................................. 60  
*M. O'Brien, D. Mortell, R. OHiggins, C McCarthy*

Modelling of novel hybrid composite-metal joints ......................................................................... 72  
*Jordan Jones, Luiz F.Kawashita, Byung Chul Kim, Stephen R.Hallett*

A mixed-mode cohesive model for delamination with isotropic damage and internal friction....... 82  
*Federica Confalonieri, Umberto Perego*

Multi-scale modeling of the structural and vibrational behavior of carbon nanotube reinforced polymeric nanocomposite plates .......................................................... 94  
*S. Doagou-Rad, J. S. Jensen, A. Islam*

Mould-in manufacturing study of thermoplastic composites with novel interlocking surface structures for adhesively bonded joints ......................................................... 106  
*John Lyons, Ronan M. OHiggins, Conor T. McCarthy*

Expanding the weight-strength-curves for unstiffened CFRP shell structures ............................... 118  
*Pawel Lyssakow, Linus Friedrich, Kai-Uwe Schröder*

Peridynamics: convergence & influence of probabilistic material distribution on crack initiation ... 128  
*Martin Rädel, Anna-Janina Bednarek, Jochen Schmidt, Christian Willberg*

Multiscale fiber kinking: computational micromechanics and a mesoscale continuum damage mechanics model ............................................................... 153  
*Miguel Heráiz, Andrew C. Bergan, Carlos González, Claudio S. Lopes*

Impact response of thick composite structures ............................................................................. 164  
*Niels van Hoorn, Christos Kassapoglou, Wouter van den Brink*

A comparative study on the modeling of matrix cracks in frp plies ............................................. 176  
*Ershad Pourbahaaddini Zarandi, Philip Simonsson, Martin Fagerstrm, Renaud Gutkin*
Monitoring of fatigue damages in adhesively bonded joints ........................................ 188
Andreas Preisler, Zamaan Sadeghi, Kai-Uwe Schröder

Discrete damage modeling of initiation and propagation of matrix cracking and delamination in clamped tapered laminated beam specimens ......................................................... 200
Hari K. Adluru, Kevin H. Hoos, Endel V. Iarve

A fatigue damage model for fiber metal laminates including fiber/matrix degradation .......... 211
C. Gerendt, E. Jansen, R. Rolfes

Environmental effects on high temperature fatigue of carbon-polyamide textile composites for aircraft applications ................................................................. 223
F. Foti, M. Gigliotti, Y. Pannier, D. Mellier

Material parameter identification challenge and procedure for intra-laminar damage prediction in unidirectional CFRP ................................................................. 235
Fabio Malgioglio, Delphine Carrella-Payan, Benoît Magneville, Laszlo Farkas

Design and optimisation of a novel interlocking joining technology for composite-metal structures . 247
Michael C. Corbett, Conor T. McCarthy

Prediction of damage progression in notched tensile specimens: comparison between two intralaminar damage models ......................................................... 259
Johannes Reiner, Navid Zobeiry, Reza Vaziri, Thomas Feser, Matthias Waimer, Dominik Schueler, Nathalie Toso-Pentecte

Development of a new standard for dynamic mechanical properties characterization of CFRP ........ 271
T. Schmack, T. J. J. Jansen, T. Filipe, F. Walther
ANALYTICAL AXIAL HARMONIC RESPONSE OF A COMPOSITE BONDED LAP JOINT

Georges E. Challita*, Ramzi A. Othman†

* Team MMC, CRSI, Lebanese University, Faculty of Engineering, Campus Roumieh, Mount of Lebanon, Lebanon
Georges.challita@ul.edu.lb

† Mechanical Engineering Department, Faculty of Engineering, King Abdulaziz University, P.O. Box 80248, Jeddah 21589, Saudi Arabia.
Rothman1@kau.edu.sa

Key words: Harmonic response; adhesive; composite; bonded joint.

Summary: A closed-form analytical solution of an axially harmonically loaded composite single lap joint with viscoelastic adhesive behavior is established. The model is based on an improved shear-lag model. The structure is fixed-free and made from two identical substrates. The whole structure was considered in the formulation; not only the bonded region. The model was numerically validated using Finite Element Method (FEM) through ANSYS workbench commercial software. For comparative purpose, two useful configurations of Carbon/Epoxy composite substrates were examined: a completely 0° fibers unidirectional (UD) and a balanced symmetric quasi isotropic (QI) configuration [0/90/-45/+45]ₘ. It was found that, for all adhesive parameters’ variation, the resonant frequencies for UD case were 1.6 times higher than those of QI case. Moreover, for fundamental frequency, QI structure was more sensitive towards adhesive damping than UD case and has shown remarkable drop in displacement at resonance while the inverse was found for the third natural frequency. For both substrates cases, no sensitivity of natural frequencies was found towards adhesive shear modulus and joint thickness. However, the overlap ratio was the main parameter influencing the natural frequencies of such structures.

1 INTRODUCTION

Adhesively bonding is imposing itself as one of the most used joining methods recently due to the light weight of the assembly and the repartition of stresses along the entire bonded surface and not only at few points of the structure. In addition, this method looks to be one of the best to assemble composites since it is known that disruptions, such as drilling or machining, will lead undoubtedly to a decrease in mechanical performance of composites. However, mechanically speaking, such structures show high complexity to be investigated. Huge efforts were made since many decades to study bonded assemblies. The majority of the studies were carried out for static loadings. However, dynamic loading studies were much less than static ones, especially in analytical calculations. Saito and Tani [1] have established analytical model for free transverse vibration of single lap bonded beams for fixed-free and fixed-fixed cases. He and Rao [2, 3] have applied energy formulation to determine natural frequencies, loss factors and mode shapes for a simply supported single lap beams. The same
energy formulation was applied later by Rao and Zhou [4] to investigate and solve numerically free vibration of a fixed-fixed tubular joint. Furthermore, the adhesive interface element along with eight-node element and energy formulation approach was applied by Ko et al. [5] to solve the resonant frequencies of single lap bonded laminated plates. Yeh and You [6] have established a finite element mathematical model for single stepped composite lap joints to find the natural frequencies; they found also that frequencies and fibers’ orientation vary inversely. An experimental validation has accompanied this analytical study. In the same context of free vibration study under finite element formulation, Lin and Ko [7] to model a patched laminated joint plates.

The first works that have dealt with harmonic response of adhesively bonded structures were those of Vaziri et al. who were interested in the effect of the void on the response of a single lap joint under harmonic peeling [8], of a tubular joint under harmonic torsion [9] and under harmonic axial load [10]. Vaziri and Nayeb-Hashemi [11] have established an analytical model to predict the response of an adhesively patch composite repaired beam under harmonic peeling load and have carried out experimental tests to correct the model. Few analytical models were established for impact case. In particular, Pang et al. [12] have used the spring-mass model to derive the governing equations of single-lap composite bonded joints while Sato [13] have used the Laplace transform to solve dynamic equations and evaluate the shear stress just at the end of a half-infinite length strap joint. However, all those models have considered the classical shear-lag model of Volkersen [14] where shear stress in the substrates thickness is not considered in the formulation. The improved-shear lag model, where a first order shear deformation in the adherents is considered, was first established by Tsai et al. [15], but for static shear lap bonded joint. Challita and Othman [16] have used this latter model to apply it on a double-lap bonded joint with similar adherents subjected to harmonic axial load; all materials were considered elastic and only the bonded region was taken in account. Hazimeh et al. [17] have succeeded to repeat the work of Sato [13] but with improved shear-lag model, which gave better accuracy towards numerical validation. Almitani and Othman [18] have extended the model [16] but for viscoelastic adhesive and substrates where they have also carried out a parametric study by examining the effect on the natural frequencies of modulus, damping and thickness of both adhesive and adherents and also the effect of overlap length.

In this paper, an extension of Almitani and Othman [18] was carried out by considering the entire structure. A FEM numerical validation was carried out on ANSYS workbench for Aluminum substrates similarly to [18]. However, responses of the structure for two main useful carbon/epoxy composites substrates UD and QI, as proposed in [15] were compared. Many adhesive parameters were varied, such as loss factor, shear modulus, thickness and overlap; their influence on the drop in displacements at resonance and on the resonant frequencies was investigated.

2 THEORETICAL FORMULATION

The structure adopted in this study is the single lap joint shown in figure 1, where three regions are considered. For each region, a free body diagram is drawn in figures 2.a; 2.b and 2.c, then, Newton’s second law is applied at each portion of the region. All quantities are written in their complex forms. The complex magnitude of any quantity \( Q \) is denoted by \( \tilde{Q} \) in the frequency domain.

Moreover, we denote the following parameters:

- \( \tilde{u}_i; \tilde{u}_{ii} \): axial displacements of substrates in regions I and III respectively.
• $\vec{N}_I; \vec{N}_{III}$: axial forces per unit width of substrates in regions I and III respectively.

• $\langle \vec{u}^{(1)}_u \rangle; \langle \vec{u}^{(2)}_u \rangle$: average through thickness axial displacements of substrates (1) and (2) in region II respectively.

• $\vec{N}^{(1)}_u; \vec{N}^{(2)}_u$: axial forces per unit width of substrates (1) and (2) in region II respectively.

• $E_s; G_s; \rho_s$: respectively longitudinal Young’s modulus; in-plane shear modulus; density of the substrates.

• $G_a; \eta$: adhesive shear modulus and adhesive loss factor respectively, this allows to write the complex shear modulus for viscoelastic adhesive $G_a^* = G_a(1+i \eta)$.

• $A_I; A_{II}; A_{III}; B_I; B_{II}; B_{III}; \alpha$ and $\beta$: integration constants function of $\omega$ calculated from boundary and continuity conditions.

Figure 1: Annotated single-lap joint

Figure 2: Free-Body-Diagram of adherents and adhesive
In addition, the following constants, defined in [18], will be used in this work:

\[
\mu^* = \frac{2G_s^*}{t_s t_s E_s} ; \quad \lambda^* = 1 + \frac{2t_s G_s^*}{3t_s E_s} ; \quad \xi^{*2} = \frac{\rho_s \omega^2}{E_s} ; \quad \zeta^{*2} = \frac{\mu^*}{\lambda^*} - \xi^{*2}
\]

Applying Newton’s second law and then Hooke’s law for fig.2a one obtains respectively:

\[
\frac{d\tilde{N}_I}{dx} = -\rho_s t_s \omega^2 \tilde{u}_I \quad (1)
\]

\[
\tilde{N}_I = E_s t_s \frac{d\tilde{u}_I}{dx} \quad \text{and then differentiating one gets}
\]

\[
\frac{d^2\tilde{N}_I}{dx^2} = E_s t_s \frac{d^2\tilde{u}_I}{dx^2} \quad (2)
\]

Replacing (2) in (1) one gets:

\[
\frac{d^2\tilde{u}_I}{dx^2} + \frac{\rho_s}{E_s} \omega^2 \tilde{u}_I = 0
\]

and solving for \(\tilde{u}_I\) then replacing in Hooke’s law one obtains \(\tilde{N}_I\) as follows:

\[
\tilde{u}_I (x, \omega) = A_I (\omega) e^{-i\xi^{*}x} + B_I (\omega) e^{+i\xi^{*}x} \\
\tilde{N}_I (x, \omega) = i\xi^{*} E_s t_s [-A_I (\omega) e^{-i\xi^{*}x} + B_I (\omega) e^{+i\xi^{*}x}]
\]

Repeating the same steps for region III (fig.2b), one might obtain \(\tilde{u}_{III}\) and \(\tilde{N}_{III}\):

\[
\tilde{u}_{III} (x, \omega) = A_{III} (\omega) e^{-i\xi^{*}x} + B_{III} (\omega) e^{+i\xi^{*}x} \\
\tilde{N}_{III} (x, \omega) = i\xi^{*} E_s t_s [-A_{III} (\omega) e^{-i\xi^{*}x} + B_{III} (\omega) e^{+i\xi^{*}x}]
\]

The expressions of average displacements and axial forces for both substrates could be directly deduced from reference [18] as following:

\[
\tilde{N}_{II}^{(1)} (x, \omega) = \frac{1}{2} i\xi^{*} E_s t_s [-A_{II} (\omega) e^{-i\xi^{*}x} + B_{II} (\omega) e^{+i\xi^{*}x}] - \frac{1}{2} \xi^{*} E_s t_s [-\alpha (\omega) e^{-\xi^{*}x} + \beta (\omega) e^{+\xi^{*}x}]
\]

\[
\tilde{N}_{II}^{(2)} (x, \omega) = \frac{1}{2} i\xi^{*} E_s t_s [-A_{II} (\omega) e^{-i\xi^{*}x} + B_{II} (\omega) e^{+i\xi^{*}x}] + \frac{1}{2} \xi^{*} E_s t_s [-\alpha (\omega) e^{-\xi^{*}x} + \beta (\omega) e^{+\xi^{*}x}]
\]

\[
<\tilde{u}_{II}^{(1)}>(x, \omega) = \frac{1}{2} [A_{II} (\omega) e^{-i\xi^{*}x} + B_{II} (\omega) e^{+i\xi^{*}x}] - \frac{1}{2} [\alpha (\omega) e^{-\xi^{*}x} + \beta (\omega) e^{+\xi^{*}x}]
\]

\[
<\tilde{u}_{II}^{(2)}>(x, \omega) = \frac{1}{2} [A_{II} (\omega) e^{-i\xi^{*}x} + B_{II} (\omega) e^{+i\xi^{*}x}] + \frac{1}{2} [\alpha (\omega) e^{-\xi^{*}x} + \beta (\omega) e^{+\xi^{*}x}]
\]

The next step is to determine the eight constants by writing the following eight boundary and continuity conditions:

(i) \(\tilde{u}_I (0, \omega) = 0 \quad ; \quad (ii) \tilde{N}_I (a, \omega) = \tilde{N}_{II}^{(2)} (a, \omega) \quad ; \quad (iii) \tilde{u}_I (a, \omega) = <\tilde{u}_{II}^{(2)}> (a, \omega) > \quad ; \quad (iv) \tilde{N}_I (a, \omega) = 0 \quad ; \quad (v) \tilde{N}_{II} (a + l_a, \omega) = \tilde{N}_{II}^{(1)} (a + l_a, \omega) \quad ; \quad (vi) \tilde{u}_{III} (a + l_a, \omega) = <\tilde{u}_{II}^{(1)}> (a + l_a, \omega) > \quad ; \quad (vii) \tilde{N}_{II} (a + l_a, \omega) = 0 \quad ; \quad (viii) \tilde{N}_{III} (2a + l_a, \omega) = \tilde{P}_0 (\omega)
\]

By expanding the latter conditions, a linear system will be established, allowing solving and calculating the eight unknowns as follows:
\[
\begin{bmatrix}
A_f \\
B_f \\
A_{II} \\
B_{II} \\
A_{III} \\
B_{III} \\
\alpha \\
\beta
\end{bmatrix} = \begin{bmatrix}
1 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\
-2K_1 & 2K_2 & K_1 & -K_2 & 0 & 0 & -K_3 & K_4 \\
-2K_1 & -2K_2 & K_1 & K_2 & 0 & 0 & k_3 & k_4 \\
0 & 0 & -K_1 & K_2 & 0 & 0 & -K_3 & K_4 \\
0 & 0 & -K_5 & K_6 & 2K_5 & -2K_6 & -K_7 & K_8 \\
0 & 0 & -K_5 & -K_6 & 2K_5 & 2K_6 & k_7 & k_8 \\
0 & 0 & -K_5 & K_6 & 0 & 0 & K_7 & -K_8 \\
0 & 0 & 0 & 0 & K_9 & -K_{10} & 0 & 0
\end{bmatrix}^{-1} \begin{bmatrix}
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
P
\end{bmatrix}
\]

Where:
\[
K_1 = e^{-iz^a} \quad K_2 = e^{i\xi^a} \quad k_3 = e^{-\xi^a} \quad k_4 = e^{i\xi^a} \quad K_3 = i\frac{z^s}{\xi^s}k_3 \quad K_4 = i\frac{\xi^s}{\xi^s}k_4 \\
K_5 = e^{-i\xi^s(2a+l_a)} \quad K_6 = e^{i\xi^s(2a+l_a)} \quad k_7 = e^{-i\xi^s(2a+l_a)} \quad k_8 = e^{i\xi^s(2a+l_a)} \quad K_7 = i\frac{\xi^s}{\xi^s}k_7 \\
K_8 = i\frac{\xi^s}{\xi^s}k_8 \quad K_9 = e^{-i\xi^s(2a+l_a)} \quad K_{10} = e^{i\xi^s(2a+l_a)} \quad P = i\frac{P_0}{E_s t_s \xi^s}
\]

The reaction \( \tilde{R}(\omega) \) at the cantilever is calculated by setting \( x = 0 \) in \( \tilde{N}_f(x, \omega) \) to get:
\[
\tilde{R}(\omega) = 2iB_f(\omega)E_s t_s \xi^s
\]

Finally, knowing all the previous quantities, we can determine the transfer function denoted by \( \tilde{H}(\omega) \) and defined as the ratio of the output, which is the axial displacement at the free end, over the input, which is the force applied at this same end. Mathematically, one can write:
\[
\tilde{H}(\omega) = \frac{\tilde{u}_{III}(2a+l_a, \omega)}{P_0(\omega)}
\]

3 NUMERICAL VALIDATION

A 3D FEM simulation through ANSYS Workbench software was carried out. The substrates were taken similarly to the study done in [18], which are made from 2 mm thick Aluminum plates. An Araldite 2031 epoxy adhesive was considered [17]. As suggested automatically by ANSYS, the mesh type generated is hexahedral with 1mm mesh size after a convergence study. As boundary conditions, the left extremity of the upper adherent is cantilevered while the right extremity of the lower adherent is subjected to a unit axial tensile force. Figure 3 shows respectively the model and the mesh.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>( E_s )</td>
<td>69</td>
<td>GPa</td>
</tr>
<tr>
<td>( G_s )</td>
<td>25.746</td>
<td>GPa</td>
</tr>
<tr>
<td>( \rho_s )</td>
<td>2800</td>
<td>Kg/m³</td>
</tr>
<tr>
<td>( E_a )</td>
<td>0.357</td>
<td>GPa</td>
</tr>
<tr>
<td>( G_a )</td>
<td>1200</td>
<td>GPa</td>
</tr>
<tr>
<td>( \rho_a )</td>
<td>0.1</td>
<td>Kg/m³</td>
</tr>
<tr>
<td>( \eta )</td>
<td>0.1</td>
<td>-</td>
</tr>
<tr>
<td>( t_a )</td>
<td>50</td>
<td>mm</td>
</tr>
<tr>
<td>( l_a )</td>
<td>50</td>
<td>mm</td>
</tr>
<tr>
<td>( a )</td>
<td>2</td>
<td>mm</td>
</tr>
<tr>
<td>( t_s )</td>
<td>50</td>
<td>mm</td>
</tr>
</tbody>
</table>

Table 1: Example of the construction of a table.

Graph of figure 4 shows a comparison between the transfer function obtained from...
numerical simulation and the transfer function predicted by the analytical model. A good agreement between both curves could be deduced, especially towards a high accuracy for natural frequencies evaluation. It should be noticed that the few oscillations observed in the numerical model could be explained by the fact that in reality deformations are not only occurring in the axial direction although the dominant displacement is always in this direction, this aspect is not taken in account in the analytical model. On the other hand, and as stated in [18], the substrates stiffness to adhesive stiffness ratio is high for Aluminum; also this is valid for composite cases which constitute the target of this study; this condition contributes in the accuracy between the two models.

4 RESULTS AND DISCUSSIONS

The main concern in this section is to apply the established analytical model for a
comparative purpose between two main configurations of Carbon/Epoxy substrates which are highly used in the industry. It should be noticed that Tsai et al. [15] have used those two configurations when establishing the improved shear lag model. The volumetric fraction of fibers is 60%. The density of this composite is 1520 Kg/m³ applying mixing law.

The first configuration consists of 8 plies, each 0.125 mm thickness, and all having 0° fibers orientation; this is a perfectly unidirectional substrate, denoted by UD (E_{xx} = 134 GPa; G_{xy} = 4.2 GPa). The second consists of 8 plies, each 0.125 mm thickness, and stacked in the following order [0/90/-45/+45]_s hence the resultant substrate will be a quasi-isotropic material, denoted by QI in this study (E_{xx} = 50 GPa; G_{xy} = 19.35 GPa). The adhesive remains the same as in the previous section.

As a first step, the comparison was carried out towards transfer function and reaction at the support. The results are depicted in graphs of figures 5 and 6 respectively.

Figure 5 shows that the first three resonant frequencies for UD structure are about 14 KHz; 45 KHz and 76 KHz while the values obtained for QI structure are 8.8 KHz; 28 KHz and 48 KHz. For each mode, the UD-frequency is greater than the QI-frequency by 1.6 times approximately. It should be noticed that the longitudinal stiffness of UD is much greater than QI while the inverse is observed for shear stiffness; this means that the longitudinal Young’s modulus of the substrate has the main influence on the natural frequency values. The low value of Young’s modulus of QI is reflected also by a higher level of displacement as shown in the graph. For figure 6, and far from resonance, the axial reaction at the cantilever \( \tilde{R}(\omega) \) is expected to be equal to the unity since the input force is also unit and this for both cases.

Another aspect to investigate is the influence of the loss factor of the adhesive on the amount of displacement drop at resonance. The values of resonant frequencies were found to be quasi-insensitive towards variation of loss factor in a certain range. Let X the resonant displacement for a very low amount of damping, \( \eta = 0.001 \) and X_0 the displacement for higher values of damping: 0.01; 0.1; 0.25; 0.5 and 1. Graphs of figures 7a, b and c show the variation of the ratio X/X_0 with respect to \( \eta \).

It is obvious that the drop increases with amount of damping for all modes and for both materials. However, one can remark that for fundamental frequency, QI is more sensitive...
to damping effect; for higher frequencies, the sensitivity of UD increases and becomes highly remarkable at the third resonant frequency. For instance, for a 0.1 loss factor, the displacement drop for UD increases from 3.8 to 5 to 13 from mode to mode while for QI it decreases from 6 to 5 to 3.

Furthermore, the variation of the adhesive shear modulus $G_a$ (GPa) = 0.214; 0.357; 0.71; 1.31 and 1.6 [18] and then the adhesive thickness $t_a$ (µm) = 20; 50; 100; 250 and 500 one can remark from graphs of figures 8 and 9 that the resonant frequencies are insensitive to those two parameters, the highest variation for all modes and both substrates materials does not exceed 1.5% towards $G_a$ variation and 3.5% towards $t_a$ variation.

![Figure 7a: Influence of damping factor on mode 1](image1)

![Figure 7b: Influence of damping factor on mode 2](image2)
Finally, a very important parameter related to the adhesive should be examined towards resonant frequencies: it is the overlap ratio, which defines the percentage of bonded length among the total length of the substrate. Mathematically, it is the ratio of the overlap length $l_a$ over the total length $a+l_a$ of the substrate. The results are depicted in figure 10 that shows a remarkable influence of this parameter on the resonant frequencies for both UD and QI configurations. Indeed, by increasing the bonded zone, the adherence between different components in the structure increases and hence the overall stiffness increases which leads to an increase in the resonant frequencies. This increase could reach the 45% between overlap ratios of 20% to 80%.

It should be noticed that although many parameters were changed, it could be remarked that the frequencies of UD case remain always 1.6 times greater than those of QI case for
any mode and any changed parameter.

![Figure 9: Influence of adhesive thickness on resonant frequencies](image1)

![Figure 10: Influence of overlap ratio on resonant frequencies](image2)

5 CONCLUSION
A closed-form analytical model based on improved shear lag model was established for a fixed-free single lap joint under axial harmonic load. This model was validated through finite element model by simulation. Two types of composite substrates were compared: UD carbon/epoxy and a balanced symmetric quasi-isotropic carbon/epoxy composite; the first has greater longitudinal Young’s modulus while the second has highest shear modulus. It was found that Young’s modulus has the highest influence since the resonant frequencies were greater by 1.6 times for UD case.
Moreover, many adhesive parameters were investigated: the amount of damping show  
high sensitivity of QI at fundamental frequency towards displacement drop while the  
sensitivity of UD was highly remarkable for higher frequencies. The resonant frequencies  
were insensitive towards adhesive shear modulus and adhesive thickness while the  
overlap ratio and the resonant frequencies vary in same way and the increment could  
reach 45% between small and high overlap ratios.

ACKNOWLEDGMENTS

The authors are thankful to Mr. Mohammad Omer and Mr. Elias Salloum for the helpful  
details they provided for the numerical side of this work.

REFERENCES

Engineering, 5*(8), 983-993, 1995.
bonded single-lap joints with a void subjected to harmonic peeling loads. *Proceedings of  
the institution of mechanical engineers, Part K: Journal of multi-body dynamics, 215*,  
subjected to a harmonic torsional loading. *Proceedings of the institution of mechanical  
subjected to a harmonic axial load. *International Journal of Adhesion and Adhesives, 22*,  


**DCB AND ENF MULTI-SCALE SIMULATIONS**

Tilen Ceglar\(^1,\star\), Martin Schwab\(^1\), Heinz E. Pettermann\(^1\)

\(^1\) Institute of Lightweight Design and Structural Biomechanics, Vienna University of Technology, Austria

* tceglar@ilsb.tuwien.ac.at

**Keywords:** Fabric laminates, delamination, embedding approach

**Summary:** Nonlinear Finite Element Method simulations of the delamination in fabric laminates with complex topology are carried out where material and interface nonlinearities are taken into account. A multi-scale embedding approach is presented which is capable to capture the nonlinear mechanisms while the computational effort is kept relatively low. The approach is used to simulate the response of Double Cantilever Beam tests and three point End Notch Flexure tests. Delamination is evaluated in terms of nonlinear load displacement curves, critical energy release rates, and dissipated energies. Besides the global response, local effects are investigated and the localized patterns of various nonlinear mechanisms are predicted. This way, detailed insight to the global delamination behavior as well as the locally involved mechanisms is obtained quantitatively.
1. INTRODUCTION

Composite laminates are widely used in many applications due to their exceptional mechanical properties. Recent developments in the experimental characterization and computational simulation of composite materials lead to improved performance of complex structures with respect to their load carrying capacity. In order to exploit the full potential of such laminates, it is necessary to develop and improve modeling approaches and computational methods which are capable of predicting their nonlinear mechanical behavior under various load cases. This is especially true for fabric laminates, which exhibit complex hierarchical structures and their response is determined by the interaction of constituents at various length scales.

Beyond the elastic behavior, all laminates without any reinforcement in the thickness direction are prone to interlaminar fracture or delamination which leads to significant degradation of stiffness and strength of the laminate. Finite Element Method (FEM) simulations of damage and failure in laminates require considerable computational power, in particular when material and interface nonlinearities are to be taken into account at high resolution. Conventional modeling strategies make use of continuum finite elements for ply constituents and zero-thickness interfaces which are applied between them. Such an approach has its limitations when it comes to modeling fabric topologies on a larger scale, where the number of degrees of freedom (DOF) quickly rises above a manageable number with respect to the computational effort.

To overcome the computational requirements of conventional three-dimensional continuum element modeling, a multi-scale embedding approach is developed here. It is based on shell element formulation, which is sufficiently detailed to capture the nonlinear mechanisms at the interfaces of fabric laminates. At the same time the computational effort is kept small enough to be handled by standard computer hardware. The multi-scale embedding approach is employed to predict the delamination process in multilayer fabric laminates. Simulations of the three point End Notched Flexure (ENF) and Double Cantilever Beam (DCB) test setups, following the standards DIN EN 6034 and 6033, respectively, are realized using the commercial FEM package Abaqus/Standard 2016 (Dassault Systèmes Simulia Corp., Providence, RI, USA).

2. MODELING APPROACH

The proposed modeling for laminates with layers of reinforcing fabric follows an multi-scale embedding approach. Microstructure with a detailed representation of the constituents in a fabric composite is embedded in a conventional shell model of the laminate at the region of interest. For the purposes of defining the microstructure, a laminate can be considered at four hierarchal length scales, as illustrated in Fig. 1.

The smallest is the tow level at which the impregnated unidirectional (UD) fiber bundles are described by a transversely isotropic constitutive law. At ply level the weaving pattern of tows embedded in the matrix is modeled by shell elements only. Also the regions of unreinforced matrix, i.e. matrix pockets, are discretized by shell elements with corresponding thicknesses. Figure 2 illustrates how the fabric composite is being assembled at ply level. At the regions where tows overlay each other they are coupled with cohesive zone elements. The laminate
Figure 1. Highly resolved fabric embedding shell model with corresponding length scales.

Figure 2. 3D illustration of the shell element based modeling at the ply level. The fabric composite (top) is disassembled into tows (left) and matrix (right) reference planes for a 2/2 Twill weave pattern [2].

... level is modeled by stacking a number of such plies with cohesive zone element in-between to model the interfaces and to enable delamination. Individual plies are stacked without any horizontal shifting, which is referred to as the in-phase stacking. To this end, the entire fabric microstructure is resolved, for more details see [1].

At component level this highly detailed representation is embedded in a conventional shell model of the laminate. For the latter, the linear elastic properties are obtained by homogenization of the above laminate scale model in a unit cell approach [2]. Hence, the corresponding shell reference planes of the conventional model are referred to as the homogenized shells throughout this paper. The embedding of the microstructure is accomplished with node-to-surface and surface-to-surface kinematic tie constrains. Consequently, the position, number, and element mesh density of homogenized shells must be chosen accordingly for each application this approach is employed. This way, the effects of the embedding on the stress distribution within the microstructure can be minimized. With such approach, the nonlinear mechanisms are restricted to the microstructure only, while the surrounding homogenized shell model of the laminate enables the global response at lower computational costs.
2.1 EXAMPLE APPLICATIONS

DCB and ENF simulations of a laminate comprising six layers of fabric plies in total are considered in order to demonstrate the present multi-scale approach. Two different multi-scale models of the laminate are set up. They embed a microstructure with two and four layers of fabric plies which have one and three cohesive zone interfaces in between, respectively. They are referred to as the EMB2 and the EMB4 model. The Fig. 3 illustrates the embedding of the microstructure in the EMB4 model. Note that the arrangement of homogenized shells depends on the number of plies in the microstructure. In both models the initial delamination reaches into the microstructure to avoid perturbation effects from the embedding. In addition to the embedded models, a simple reference model is set up as conventional shell model of the laminate and homogenized materials properties. More details about these models can be found in [3].

Material. The material data used for the purposes of this work represent a carbon/epoxy composite with 2/2 twill weave fiber architecture. The corresponding properties of each constituent in the model can be found in Table 1. The tows are modeled as linear elastic, transversely isotropic, whereas the unreinforced matrix pockets are represented by linear elastic, isotropic material. The cohesive zone elements representing the interface between individual plies include a bilinear traction separation law with quadratic nominal stress damage initiation criterion and energy based damage evolution. In Table 1, $t_{i,max}$ are the interface strength values, $G_{ic}$ are the critical energy release rates and $K_i$ denote the initial linear elastic interface stiffness. The indices I, II and III denote the three modes of delamination, i.e. opening, sliding, and tearing mode. The mixed-mode response is prescribed with the Benzeggagh-Kenane criterion [4], where the parameter $\eta$ is defined. Note that the cohesive zone elements coupling overlaying tows in the plies have linear elastic properties, thus only interface stiffness, $K_{ic}$, is applied to them.

![Figure 3](image-url)  
Figure 3. Sketch of a cross section of the EMB4 model indicating the homogenized shell reference planes and embedding of the four layer microstructure. Shell thickness are illustrated by hatched areas, circles represent featured edges (not nodes). Figure not to scale.
**Geometry and boundary conditions.** The laminate’s geometry is defined in such way that it enables a straightforward embedding of the textile microstructure. It is placed around the initial delamination front as shown in Figs. 4 and 5, where the load and boundary conditions are also illustrated, following the scheme of the ENF and DCB test setups. Note that the initial crack length, \(a_0\), is longer in the models for the ENF simulation compared to the DCB simulation to ensure a stable crack growth [5]. In the ENF test simulations, frictionless contact formulation with penalty stiffness is applied between the surfaces of delaminated plies.

Table 1. Material properties of the carbon/epoxy 2/2 twill weave composite and its constituents.

<table>
<thead>
<tr>
<th></th>
<th>Tows</th>
<th>Homogenized ply</th>
</tr>
</thead>
<tbody>
<tr>
<td>(E_1) (MPa)</td>
<td>142176.97</td>
<td>(E_1^h = E_2^h) (MPa)</td>
</tr>
<tr>
<td>(E_2) (MPa)</td>
<td>13819.56</td>
<td>(G_{12}^h = G_{13}^h = G_{23}^h) (MPa)</td>
</tr>
<tr>
<td>(G_{12} = G_{13} = G_{23}) (MPa)</td>
<td>6251.81</td>
<td>(\nu_{12})</td>
</tr>
<tr>
<td>(\nu_{12}) (=) (\nu_{23})</td>
<td>0.23</td>
<td>(0.045)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Matrix</th>
<th>Ply interface</th>
</tr>
</thead>
<tbody>
<tr>
<td>(E_{\text{Matrix}}) (MPa)</td>
<td>3250</td>
<td>(t_{\text{I, max}}) (MPa)</td>
</tr>
<tr>
<td>(\nu_{\text{Matrix}})</td>
<td>0.37</td>
<td>(t_{\text{II, max}} = t_{\text{III, max}}) (MPa)</td>
</tr>
</tbody>
</table>

**Tow interface stiffness**

\(K_1 = K_{\text{II}} = K_{\text{III}}\) (N/mm\(^3\)) \(= 10^3\)

\(K_1 = K_{\text{II}} = K_{\text{III}}\) (N/mm\(^3\)) \(= 10^5\)

\(G_1 = G_{\text{II}} = G_{\text{III}}\) (N/mm) \(= 2.0\)

\(G_{1c} = G_{\text{IIc}} = G_{\text{IIIc}}\) (N/mm) \(= 2.0\)

\(\eta\) \(= 2\)

![Figure 4. Geometry and boundary conditions of the models used in the ENF simulations.](image)

![Figure 5. Geometry and boundary conditions of the models used in the DCB simulations.](image)
3. RESULTS AND DISCUSSION

First, the results from the ENF simulations are compared for the reference, EMB2, and EMB4 models. Their global response is evaluated in terms of load displacement curves. Additionally to the numerical predictions, an analytical solution is given by the Corrected Beam Theory (CBT) [6] for the corresponding interface and homogenized material properties. The main focus of investigation is the nonlinear response during delamination. For this reason, relation between average crack length and central displacement are examined in details. The average crack length is evaluated by means of counting the completely damaged cohesive zone elements in the interface and thus gives an averaged value over the width of the laminate. Any attributes of a crack, e.g. length, growth or propagation, discussed in this work relate to the numerical representation of a crack explained above. The average crack length is than correlated with the propagation of the fracture process zone through the interface. The fracture process zone refers to the cohesive zone elements exhibiting softening behavior. This way, a detailed insight to the locally involved mechanisms is obtained. Finally, delamination is evaluated in terms of energy release rates and dissipated energies. Simulation results from the DCB test setup are evaluated in the same way, however, for the sake of brevity only load displacement curves are presented in this paper.

Displacement controlled nonlinear FEM analysis of delamination in the present multi-scale models require stabilization control in order to achieve converging solutions. For this purpose, viscous regularization in the constitutive law of the cohesive zone interfaces is applied with a relaxation time of $10^{-6}$ s.

**ENF simulations.** Load displacement curves of the ENF simulations are shown in Fig. 6. The enlarged view in the figure shows the response during delamination, where the embedded models exhibit highly non-linear behavior. This implies that delamination process in fabric laminates is not a steady state phenomenon, as the reaction force decreases in a quasi step-wise manner.

The average crack length in relation to the central displacement is presented on Fig. 7 for all models. The enlarged view in this figure shows a typical quasi step in the crack propagation with annotation points at significant transitions in the response. These points are correlated to Fig. 8, where the fracture process zone is presented at the middle interface of the EMB4 model. The integration points in cohesive zone elements are evaluated with respect to the damage initiation criterion, i.e. the quadratic nominal stress criterion. Completely damaged cohesive zone elements are hidden in order to illustrate the crack front. Note that on the crack front the integration points are already completely degraded.

The fracture process zone develops with a high dependency on the reinforcement topology adjacent to this interface. This is clearly shown by shape of the damage initiation front through the width on Fig. 8. At the interface which is coupled to the matrix pockets oriented in the direction of delamination progress, the damage initiates with a noticeable delay. At ply level, the local stiffness changes throughout the ply. The matrix pockets have significantly smaller
stiffness compared to the tows due to their absence of reinforcement phase. Consequently, the matrix pockets deform more compared to the surrounding reinforced structure. On a laminate level this means that a smaller share of the external energy is invested into deforming the interfaces coupled to these regions.

The same local mechanism also governs the propagation of the fracture process zone through the interface. Examining the correlation between the average crack length at the notation points in Fig. 7 and the fracture process zone sequence in Fig. 8 gives a reasonable interpretation. At point (a) the crack front is, in average, at the edge of matrix pockets which are oriented parallel to the crack front. From point (a) to point (b) there is a relatively small crack extension, see Fig. 7. In this phase the crack propagates through the area of matrix pockets. At point (b) the crack front reaches an area, which is reinforced through the whole width of the laminate. Here the local stiffness in adjacent plies increases and consequently the crack propagates faster until point (c), where the damage initiation front again reaches the area with parallel aligned matrix pockets. There the delamination process is slowed down again. This process is repeated through the interface as shown in Fig. 7. Note that at crack lengths above 65 mm the fracture process zone in the multi-scale models gets close to the edge of the middle interface in the microstructure and results are no longer reliable.

Figures 6 and 7 also show, that delamination onset in the EMB4 model is predicted at lower central displacement compared to the EMB2 and the reference model. Note that the initial crack front in the embedded models is placed at the region which is reinforced throughout the whole width of the laminate. In the EMB4 model, where four layer of fabric are modeled in
the microstructure, the local stiffness at this region is higher compared to the same region in the EMB2 model, hence the critical energy release rate in the interface is reached at lower central displacement compared to the other models.

In the EMB4 model, the influence of delamination in the middle interface on the damage process in the adjacent ones can be investigated. Figure 9 shows the localized fracture process zones that are predicted in both adjacent interfaces. From the damage initiation criterion in both interfaces it can be seen that the damage initiates at locations, where the individual interface is coupled to matrix pockets from both sides. The elements at these regions exhibit some softening, however the stiffness degradation remains small at these integration points, see damage evolution criterion in Fig. 9. Therefore no crack openings or significant damage in these interfaces are predicted by the present model.

The Fig. 10 (a) shows the comparison of the models in terms of energy release rates during delamination. The simulation results are evaluated by employing CBT. First note that this evaluation is based on a simplified laminate model and thus only gives approximate values for the multi-scale models. Annotation points are correlated with previous figures. Between points (b) and (c) the evaluated energy release rate has a slight negative slope ($\frac{\partial G}{\partial a} < 0$) which indicates an unstable crack growth. In fact, here is the crack propagating the fastest with respect to the central displacement, see Fig. 7. The difference to the critical energy release rate, $G_{IIc}$, is due to the evaluation approach of the CBT. Figure 10 (b) shows the dissipated energy from FEM predictions due to damage for the whole model, denoted to as ALLDMD. The critical value is marked with a dashed line. The slope of the curves represent the energy release rate. The difference between the ALLDMD energy output and energy evaluated by CBT lies in the fracture process zone, where the partly damaged integration point contribute to the dissipated energy but are not included in the CBT evaluation.
Figure 8. Fracture process zone sequence in the EMB4 model during the ENF simulation. Subfigures are correlated with the notation in Fig. 7. Damage initiation variable QUADSCRT is shown on the cohesive zone elements, see bilinear traction separation law on the right. Completely damaged elements are hidden. The weaving pattern of ply below the interface is illustrated with outlined rectangles.

Figure 9. Damage initiation and onset in the top and bottom interfaces of the EMB4 models. The average crack front in the middle interface is at point (c) which is illustrated with a dashed line, see Fig. 7. Delaminated interface is left of the line.
Figure 10. Dissipated energy during ENF simulations. The left figure shows the energy release rates during delamination evaluated by CBT. In the right figure, the dissipated energy due to damage in the whole model is compared to the CBT evaluation. Dashed lines illustrate the defined critical energy release rate in the cohesive interfaces.

**DCB simulations.** Figure 11 shows the load-displacement curves for the DCB simulations of the EMB2 model. The difference in the response prior to delamination onset is due to the formulation of the fracture process zone, where the cohesive interface is being deformed and thus the global stiffness of the laminate is gradually decreasing to the point of delamination onset. The delamination process is evaluated with the same approach as in the ENF simulations. The fracture process zone is, in average, smaller due to the lower critical energy release rate for mode I, however the delamination process is influenced by the same local mechanisms as predicted and evaluated in the ENF simulation.

### 4. CONCLUSIONS

A multi-scale embedding modeling based on a shell element formulation is applied to model multilayer fabric laminates and to predict their nonlinear response during standard delamination test setups. Exceeding the global response, local effects in the microstructure are taken into account and localized patterns of the various nonlinear mechanisms are predicted. The response is also evaluated in terms of dissipated energies attributed to these local mechanisms which eventually determine the overall dissipation. The development and propagation of the fracture process zone in the interface show a clear dependence on the topology of plies adjacent to it. Moreover, effects of delamination in one of the interfaces on damage in the adjacent ones are predicted. Therefore, the approach proves itself as being numerically efficient and it enables detailed insight to the nonlinear behavior fabric laminates exhibit during delamination.
Figure 11. Load displacement curve for the DCB simulation. The enlarged view shows the nonlinear response during delamination.

References

INTERLACED COMPOSITES WINDING (D-3F TECHNOLOGY) AS AN ALTERNATIVE FOR BRAIDING AND WINDING

Daniel Denninger¹, Maik Berger¹, Yordan Kyosev²

¹Technical University Chemnitz, Professorship of Assembly and Handling Technology, Reichenhainer Straße 70, 09126 Chemnitz, Germany
daniel.denninger@mb.tu-chemnitz.de

²Hochschule Niederrhein, Faculty of Textile and Clothing Technology, Research Institute of Textile and Clothing, Webschulstrasse 31, 41065 Mönchengladbach, Germany
vordan.kyosev@hs-niederrhein.de

Key words: Rotary braiding system, covering, weaving-like, interlacements

Summary

This paper deals with the development of a new high-performance fiber (carbon or glass) processing technology called D-3F. There are well-known textile technologies for the production of profiles or covered tubular cores in order to increase their radial stiffness, like composite winding. The main disadvantage of wound preforms is the limitation in stability of the single windings, caused by missing interlacements. An advantage might be the fiber deflections and the resulting directed mechanical properties in these preforms. On the other hand, it is impossible to cover profiles with changing diameters and resulting local angles higher than the friction angle of the current materials. These forming profiles are often covered by the more suitable braiding technology. Braided structures can be produced stably based on proper interlacement between the two or three thread systems inside.

In fact, the braiding technology has its limitations concerning the productivity. The interlacement of the thread, which stabilizes the structure from one side, is the reason for broken filaments as a result of the deflection of the threads from the other side. The number of broken filaments is proportionally increasing with the speed of the braiding process. In order to combine the advantages of both composites winding and braiding into a new productive and fiber-friendly technology, a new machine concept was developed and patented. This work presents its advantages and disadvantages and gives examples of calculations about the properties of the structures based on the method of inclusions.
1 State of the art - covering processes

The idea of composite materials evolved in bionics. In a composite, different materials are combined in a purpose-oriented manner in order to obtain improved or new macroscopic properties against the starting constituents and to control undesirable side effects. Humankind recognized the outstanding potential of fiber composite materials as a construction element from early on. For example, already 10,000 years ago, a combination of fiber materials (straw or reeds) and matrix (clay) served as a building material.

Today, mainly lightweight construction applications require the use of high-performance fibers (such as carbon or glass) for a defined mechanical reinforcement structure. Highly viscous reaction resins serve as the matrix. In the liquid state, these allow the addition of various additives as well as the force-flow-oriented integration and alignment of the said continuous fibers. The polymerization is initiated by the addition of free-radical generators and thereby a cross-linked thermosetting plastic matrix is formed. [1]

There are established technologies for the production of profiles or covered tubular cores in order to increase their radial stiffness. The individual high-performance fibers are initially combined, as so-called rovings, and are prepared as prepregs or manufactured directly in textile processes to form contoured semi-finished fiber products or preforms. Based on the distinction mentioned above, a general classification into the two superordinate process types - the draping process and the resin transfer molding RTM - can be made. [2]

In the draping process, prepregs are processed into geometrically determined bodies. These bodies are cured under temperature and pressure to produce components. The fibers contained in the pre-impregnated rovings may be in the form of a unidirectional layer, a woven fabric or another fabric and are surrounded by the uncured thermosetting matrix (reaction resins). The main representatives for the production of profiles or coated tubular cores with prepregs are composite winding and pultrusion. [3]

In contrast to the draping process, semi-finished fiber products or preforms are in the first place manufactured for the RTM process. These textile intermediates are placed in a shaping tool, then impregnated with resin and cured under temperature and pressure. Depending on the method of preparation of the preforms, these have process-related binding structures and therefore variable mechanical properties as well as different draping and handling ability. Particularly for the production of profile-shaped or final-contoured preforms, only a few textile processes could be established [4]:

- knitting (meshed systems)
- wrapping/winding (non-mesh systems, without interlacements)
- braiding and weaving (non-mesh systems, with interlacements)

The knitting processes are not taken into account for all further. Although the knitted fabrics are distinguished by a comparatively good draping and handling ability as well as damage resistance, due to potential fiber stretching, the multiple fiber deflections and changing fiber orientations lead to proportionally decreasing mechanical properties.

In contrast, especially the multi-layered wound preforms have a constant fiber orientation per layer. These windings are characterized by minimal fiber deflections on the micro- and mesoscopic-level and can therefore be ideally force-flow-oriented to increase the mechanical
properties of a component. The main disadvantage of wound preforms is the limitation in stability of the single windings caused by missing interlacements. The draping and handling of wound preforms is therefore only ensured to a limited extent, since the individual layers tend to slip. Regarding the fiber deflections and the resulting mechanical properties in these preforms that may be an advantage. On the other hand, it is impossible to cover profiles with changing diameters and in the same local angles higher than the friction angle of the current materials.

Basically, only comparatively short and easy-to-handle bodies can be wound. There are a number of different methods for circumferential or cross-winding with rotating or stationary cores, a well-known example is the production of high-pressure tanks [5]. To increase the productivity, wide taps or even a series of laying heads can be used. The main problem is the mentioned slipping of the layers, which occurs from a critical angle with changes in the cross-section. This geodetic angle is dependent on the prevailing friction conditions, the selected thread tension and the entire profile geometry. For a more detailed look at the winding process, please refer to the relevant technical literature at this point. [2, 3, 4]

Forming cores with cross-section changes are often covered by the more suitable braiding technology. Braided or weaving-like structures can be produced stable based on proper interlacement between the two or three thread systems inside. Weaving-like structures essentially include the tissues and braids. In theory, they are also defined by a concrete number of layers. In contrast to the pure windings, these layers are interconnected by crossings (interlacements). These interlacements are the reason for significantly higher fiber deflections. With these deflections, proportionally decreasing mechanical properties with simultaneously good resistance to damage are to be expected due to potential fiber stretching and easy handling. Above all, the slipping can be largely avoided, but not completely eliminated, with the corresponding configuration of the braiding parameters. The advantage of a braided structure is relative shape stability even without a core. [6]

In particular, the braiding process offers excellent conditions for the gentle processing of high-performance fibers and for the economical serial production of load-bearing semi-finished fiber products or final-contoured preforms. The comparatively high fiber deposits of the braiding technique are ideally combined by the layer-wise overlapping of forming cores with a simultaneously large production repertoire for geometrically flexible design. For the realization of a reproducible and force-oriented fiber orientation, three thread systems are usually processed into a triaxial UD braid. [7]

Main port of conventional production processes are circular or radial braiding machines, based on the maypole system with mechanically forcible laying of threads. In fact, braiding has its limitation concerning the productivity. The interlacement of the threads, which is stabilized by the structure from one side, is the result for broken filaments. The number of broken filaments is proportionally increasing with the speed of the braiding process. [8]

Rotary braiding machines – especially lever arm braiding machines in accord with the system "Horn" – are not used. The advantage of the system "Horn" as opposed to the maypole system, is the constant tangential bobbin arrangement in combination with the characteristic laying of the threads as well as the accompanying fiber friendly and position-parallel processing of the braiding material. At the same time, the increased braiding speed, due to the machine concept with the opposite rotational movement of two rotors, is an enormous advantage for economical serial production [9].
2 Development of the interlaced composites winding technology D-3F

Objective of the following study was to extrapolate the potential of the system "Horn" for the production of semi-finished fiber products or Preforms by covering a forming core. The challenge was to integrate a third thread system into a laying technique, based on two counter rotationally rotating thread systems, which are suitable for a high braiding speed [9, 10]. For the synthesis of the machine concept, a process-oriented synthesizing method was elaborated. This method is based on a so-called technology synthesis and necessary to realize a gentle and parallel transfer of threads. The methodology’s initial point was a synthesizing method with a direct coupling to a kinematic process simulation to design the transfer of threads. The synchronization and optimization of the transfer of threads was carried out by an iterative synthesis of mechanisms in close connection with the kinematic analysis of the suggested drive-concept. In a structural design, the physical feasibility of the innovative drive-concept as well as the non-destructive transfer of threads, was established.

The technological synthesis for integrating a third thread system lead to a novel and textile-technological manufacturing process with a new weaving-like binding structure as a combination of laying and braiding or weaving, "D-3FG" ("Denninger-3Faden Geflecht"), see Figure 1. Mentioned binding structure combines the properties of two layers and weaving-like interlacements. Through the interlacements, these layers are arranged one inside the other and fixed in their respective orientation. In analogy to the braiding angle, the angle of the laying threads can be varied in some manner over the take-off speed and can thus be used for the orientation of the reinforcing fibers. Along with that, the drive-concept of the new rotor braiding system D-3F was patented (file reference 102014016832.8) with the title “Flechtvorrichtung und Flechtverfahren zum Überflechten eines Flechtkerns“ [11].

![Figure 1: D-3FG structure compared with the conventional bi- and triaxial braids](image)

The result of the logical interpretation of the new laying technique, based on two counter rotationally rotating weft thread systems, is shown in the following Figure 2. By means of the above-mentioned thread systems, two superimposed layers of oppositely oriented threads are deposited on a core. In order to avoid superimposing relative movements, the third braided thread system was arranged in a stationary manner in such a way that an interlacement of the overlaying layers can be achieved at their crossroads. The rotary braiding system D-3F matches a targeted combination of all characteristics typical for braiding in a concept that is suitable for processing Preforms with a tangential alignment of bobbins and parallel laying of threads. Thus, the Preforms are manufactured by
“overbraiding” with the new weaving-like structure. According to the patent claim, the drive-related functionalization of the novel laying technique in a circular concept is solved as follows [11]:

- “At least one first bobbin carrier set and at least one second bobbin carrier set, where at least one of the two bobbin carrier sets can be moved along a circular path, where between the two bobbin carrier sets when over-braiding the braided core a relative movement can be executed and where by the relative movement, each of the bobbin carriers of the two bobbin carrier sets drawn yarns can be stored as intersecting weft yarns to the braiding.“
- “At least one third, stationary arranged bobbin carrier set, and“
- “At least one actuating element, where the yarns that are unwound throughout the process of over-braiding the braided core from the bobbin carrier of the third bobbin carrier set by at least one adjusting element in an oscillating upward and downward movement are displaceable and thereby are laid as warp yarns to form a binding alternating above and below the crossing points of the weft yarns.”

Figure 2: rotary braiding system **D-3F**

In the proposed system concept, the weft thread bobbin carriers are fastened to slides, which are mounted on guide rails and are again mounted on a fixed slide track at a defined distance from one another. To enable the shedding of the warp threads, the slider web must be provided with slots at equidistant intervals. The openings of the opposing slots are aligned with each other, **Figure 3**. For the production of the new fabric-like binding structure, warp threads are provided by the third thread system. These warp threads are drawn off from an outer and stationary filament carrier set by an individual thread positioning element. These elements are oscillating from a lower to a higher latitude with a collision-free passage of the weft bobbin carriers and are thereby laid above or below the intersection points of the weft threads. The opening and closing of a compartment takes place in the time windows in which the slides are not guided over the slits of the guide rails.
For a synthesis of the warp thread positioning in greater detail, a kinematic process simulation was used for the continuous parameter matching of the technological, kinematic, structural and constructive requirements. This process can therefore be described as an interdisciplinary innovation process with direct linking of the textile- and the drive-technology implementation. For the basic synchronization of the warp thread positioning with the two counter rotationally rotating weft thread systems, static position specifications, resulting from determined process points, were first implemented as a drive of the thread positioning elements into the synthesis model. The mapping of the braided path, which has been described as a space curve on the braid core in the CAE/CAD system with the aid of the predetermined kinematics of the thread draw points and the braid core, carries out the first visualization of the novel binding structure. [10]

3 Design studies and possibilities of D-3F

3.1 Basic Winding options

In addition to the synthesis and simulation of the rotary braiding system D-3F, design studies have been performed to evaluate further possible binding structures and their potential applications. In particular, the variation of the movement of the warp threads permits a defined range of configurations for the binding variation of the interlaced composites winding technology. However, such design studies lead to a non-practical computational effort in the used CAE / CAD system. On these grounds, the development of an effective solution for the design and visualization of the possible binding structures became necessary.

Due to the existing functionalities for the calculation and visualization of braided products with known binding structures, the software TexMind Braider [12] was used as a basis for development. The clear analytical description of the thread paths leads to a real-time
calculation of chosen settings. In addition to the image of braided products, known binding structures can be set variably, taking the parameter configurations such as braidiness, cross-sections, number of fibers, coloring, etc. into consideration. Furthermore, the generated 3D geometries can be exported in FEM tools and processed for computations. Calculations for the output quantity, taking into account the parameter settings and other machine-specific feasibility estimates, are already implemented.

**Figure 4** shows the prototypical version of the **TexMind Braider D-3F**, based on the conventional calculation core. This toolbox already provides all necessary functionalities for the design of theoretically possible binding structures with three thread systems. This allows the mapping of the whole bandwidth of the interlaced composites winding technology by neglecting machine-specific restrictions. Therefore, the implementation of the binding variables is built up systematically, using a matrix-based input. The superimposed layers of oppositely oriented weft thread systems are generally present (Figure 4: red, green).

![Figure 4: using the new toolbox in TexMind Braider for design studies of the textile structures](image)

In principle, an interlacement with the aid of a warp thread from the warp thread system is possible at each intersection of these opposing threads. Each column of the matrix corresponds to a series of intersections in a horizontal arrangement. By filling the matrix with digits, the binding structure takes place over a defined number of rows according to the following definition:

- 0 means no interlacement
- 1 means the interlacement of one warp thread system
- 2 means the interlacement of both warp thread systems.
The structure of the rotary braiding system D-3F allows the fabrication of a variety of weaving-like binding structures as well as the winding or spiralling of a guided core with rotating warp thread systems (winding carriers). The weft thread system can also be used with the aid of an appropriately fixed position of the thread, laying units for the removal of non-interlaced 0 degree threads as shown in Figure 5. Consequently, the following general options of thread laying without interlacements are given:

- one-thread system (winding single layer with multiple carriers)
- two-thread system (winding two layers S, Z)
- two-thread system (winding one layer S or Z and 0 degree without interlacement)
- three-thread system (winding two layers S, Z and 0 degree without interlacement)

\[ \text{Figure 5: general options without interlacements based on three thread systems} \]

In addition to the shown options for the processing of three thread systems, it is generally possible to build up structures for the processing of an arbitrary number of thread systems. The applicable restrictions are derived from the technical and economic boundary conditions and their feasibility. Especially the size of the bobbins and their carrier arrangements are responsible for the kinematic and kinetic effects which are used for the constructive design. However, there is no general technological restriction.
3.2 Adding interlacements between layers

An important advantage of the rotary braiding system D-3F is the stationary arrangement of the warp thread system. In particular, this stationary arrangement permits the use of large bobbins with the appropriate thread supply for the compensation of undesirable thread length and thread tension changes. This controlled thread supply is required for high process speeds with low fiber damage due to friction-related relative movements on the thread guide elements by the laying devices.

In order to combine the advantage of windings and weaving-like structures, the two layers (Figure 5, b) have to be fixed by adding interlacements with these weft threads. General options with interlacements, based on three thread systems, are shown in the following Figure 6, Figure 7 and Figure 8, as described:

- Figure 6: One weft thread system placed on a ring is winding by rotating and one stationary warp thread system is interlacing by going up and down
- Figure 7: Two weft thread systems placed on rings are winding by rotating in opposite directions and one stationary warp thread system is interlacing both rotating systems
- Figure 8: Two weft thread systems placed on rings are winding by rotating in opposite directions and one stationary warp thread system is interlacing both rotating systems in a softer way

Figure 6: a) 3D Model with one layer interlaced with the second system completely through the thickness b) column for the structure
3.3 Possible applications and economical aspects

As the previous figures and examples demonstrate, the D-3F technology is able to cover profiles with two thread systems (or prepregs) in its basic counter rotational configuration. By using the additional warp thread system for interlacing these two layers, the advantages of filament winding and braiding can be combined. For this combination, the warp thread system has to consist of very thin connecting threads. With this method, the interlacements will not integrate countable crimp, similar to the triaxial UD braid. The interlacing threads will keep the covering structure stable and the two layers will be connected to each other.
Concerning the design of a prototype and the necessary exploration of the rotary braiding system D-3F, it is planned to apply servomotor drives for the laying devices of the warp threads. With these motion controlled systems, the frequency and the depth of the interlacements can be selected and adjusted according to the matrix-based input of the TexMind Braider D-3F. In order to keep the structure stable, the binding structure could be adjusted by building up more interlacements during the covering process of cores with conical cross-sections.

Main advantage of the rotary braiding systems is the technical implementation of the braiding process by mentioned laying devices. In contrast to the conventional maypole braiding machines, the bobbin carriers – which have a high and dynamically moving mass – are only performing in a constant rotational motion. The necessary changes of the moving direction for the production of interlacements are carried out by laying devices. For that reason, the rotary braiding machines can show a significantly higher productivity rate than conventional maypole braiding machines.

Another advantage to be mentioned is the constant tangential bobbin arrangement. The rotary braiding system D-3F was developed especially with a focus on the arrangement and space between the carriers. Thereby, lager tapes or heavy tows could be used as weft threads. Therefore, the system is designed to achieve an economically and technologically effective machine to cover small and medium diameters of composite profiles for pultrusion or later lamination. This advantage is accompanied by a significant disadvantage. The system D-3F has a more stringent limit on its potential size. The braiding machine D3F with its current conceptual design and constant tangential carrier arrangement would, as a consequence, not be ergonomic, if it has to handle 48 or more carriers. That is why the overbraiding of thicker profiles will probably remain an area of the radial braiding machines with 144, 288 and more carriers.

4 Conclusion

This paper showed the development of the interlaced composites winding technology D-3F as an alternative covering process. This alternative covering process with a new circular drive-concept to generate an innovative shifting motion is a result of the recombination of required sub-functions which are needed to manufacture the best possible and ideal binding structure. The technological synthesis for integrating a third thread system led to a novel and textile-technological manufacturing process with a new weaving-like binding structure as a combination of laying and braiding or weaving. Several structures are visualized with a new module of the TexMind Suite – named TexMind Braider D-3F. The analysis of the virtual samples shows, that the D-3F has potential in replacing composite winding in some class of composites, where the braiding diameter is not too high and where the interlacement between two layers can improve the stability of the structure. The investigation of the economical and mechanical efficiency of the D-3F machine and the resulting composite structures still have to be verified with experimental investigations, which include the realization of a prototype of the machine and the production of samples. For this upcoming task, the authors are open for cooperation.
References


HIGH SPEED MODELLING OF POLYAMID-MATRIX WOVEN COMPOSITES

Nicolas Feld*, Fabien Coussa†‡, Benoit Delattre†, Jacques Renard†

* Safran Tech
Safran Paris Saclay, rue des Jeunes Bois – Châteaufort, 78114 MAGNY-LES-HAMEAUX, FRANCE
nicolas.feld@safrangroup.com

† Groupe PSA
Centre Technique de Vélizy, route de Gisy, 7814 VÉLIZY, FRANCE
benoit.delattre@mpsacom

‡ Centre des Matériaux
Mines ParisTech, CNRS UMR 7633 , PSL Research University, BP 87, 91003 EVRY, FRANCE
{fabien.coussa,jacques.renard}@mines-paristech.fr

Keywords: Woven Composites, Computational Modelling, Dynamics

Summary: This paper proposes a new computational model for polyamid-matrix woven composites that takes into account rate effects for high speed impact applications. The rate effects are captured solely through the use of a spectral viscoelastic model, used in parallel of a conventional damageable elastic-plastic model. It appears that neither plasticity nor damage kinetics need, at first order, any rate dependency to capture the nonlinear envelope response until failure at very different loading rates. To verify this hypothesis, a dedicated experimental setup is proposed that allows strain-controlled interruption of high speed tension loadings. Cycled loading sequences may therefore be applied to a sample even in the dynamic regime, and loss of stiffness and irreversible strains may be measured until total failure. A specific and rigorous identification procedure is proposed in that effect. Our experimental findings confirm the ability of the model to predict both the envelope response curve and the evolution of the internal mechanisms at very different strain rates.

1. INTRODUCTION

Car manufacturers are looking for solutions to lighten their vehicles in order to meet pollutant emission requirements – for thermal engines – or to extend their range – for electric vehicles. Composite materials, among other solutions, offer excellent strength and durability. However increased per unit cost and manufacturing time are among the main drawbacks material producers have to overcome in order to make composite solutions viable for mass-market production. In that prospect, woven glass fibers and thermoplastic matrices respectively are more serious contenders than aeronautic-grade composites, at least on paper. Their adaptiveness to various automotive applications remains however to be proved. In this paper we study
the behavior of a glass fiber woven composite using a 2x2 twill weaving pattern and a polyamid 6,6 matrix, manufactured by thermostamping and produced by Du Pont de Nemours. The objective application is a door reinforcement module, whose main function is to act as a safety net, adding its own stiffness and strength to that of the steel door and preventing any foreign object from entering the passenger compartment. A main preoccupation is therefore the constitutive behaviour of that material under strain rates varying from $10^{-3}$ s$^{-1}$ to approximately $10$ s$^{-1}$.

Early experimental investigations reveal a very strong strain-rate sensitivity, even at low strain rates, of the material in all directions\(^1\) (see Figure 1). Stiffness is noticeably affected, mostly in the bias direction, while the effect on strength is more obvious, even in the fibers’ directions. Higher strain rates exacerbate these trends. Let us also assess the strong anisotropy; not so much in stiffness and strength, which vary by a ratio of 10 and 3 respectively, which is much less than for carbon fiber reinforced plastics. The most visible aspect of anisotropy rather concerns ultimate strains, which vary from $\approx 1.75\%$ in the fibers’ direction to $\approx 30\%$ and more in the bias direction. Yet these large strains are never observed in practical applications, since stress redistributions and multiaxiality always cause fiber failure before shear strains exceed 8 to 10%. Therefore finite strains will not be investigated in the following.

![Graphs showing the monotonous response of 0° and 45° oriented samples submitted to low strain rates](image)

Figure 1. Monotonous response of 0° and 45° oriented samples submitted to low strain rates

Few composite models take into account the rate dependency of the nonlinear response in intermediate dynamics. For good reasons, since epoxy, PEEK, and other aeronautic-grade matrices display little rate dependency in this dynamic regime. Let us briefly examine the properties of a few recent rate-dependent models suited for impact modelling.

\(^1\)Thereafter relative orientations with respect to the warp direction may be used instead of material directions to describe prescribed loadings. The 0° orientation refers to the warp direction, 90° to the weft direction, and 45° to the bias direction. Warp and weft are indistinctively referred to as the fibers’ directions, and since the weave is balanced, the behaviour is assumed and verified to be similar in both directions.
In [1], strain rate is introduced in three occurrences. The elastic moduli are strain rate dependent, but in a parametric manner – i.e. no visco-elastic model is considered; plastic flow obeys a Norton-Hoff isotropic law; and damage rate is bounded by a decreasing exponential [2]. The latter dependency has for sole objective the regularization of the softening model by preventing damage from increasing at a quasi-infinite rate once localization initiates. This aspect is retained in our study. Apart from regularization, damage is therefore considered rate-independent, while plasticity is, conversely, strongly affected. This hypothesis, in analogy with metals, is however difficult to justify a priori; and we will see that it does not apply to our material. Finally elasticity is introduced as log-dependent with strain rate. Although this is a fair approximation at first order, such formulation causes all nonlinear mechanisms to be strongly coupled with strain rate. In order to keep the identification procedure simple and robust, a different approach will be favored.

In [3], the authors consider that strain rate only affects the nonlinear behaviour beyond a certain elastic limit. They introduce a rather quirky rate dependent plastic law:

\[ \varepsilon_p = \chi (\dot{\varepsilon}_p)^m (\sigma)^n \]  

where \( \varepsilon_p \) and \( \sigma \) are plastic strains and stresses respectively, and \( \chi, m, \) and \( n \) are material parameters. Although no damage model is considered in this study, a strain rate dependent failure criterion is also introduced. For that purpose, the authors use the Monkman-Grant equation, in analogy with metals, and extend it to the anisotropic case. This approach departs strongly from a phenomenological modelling philosophy, yet the attempt to keep the number of parameters to a minimum is a preoccupation we share.

In [4], only the reversible behaviour of \( \pm 45^\circ \) laminates is studied. The authors rely on a so-called spectral viscoelastic model, usually applied to quasi-static loadings [5]. In their work, it is extended to dynamic ranges thanks to a second spectrum, covering high strain rates. The spectral viscoelastic model has the benefit of covering a wide range of characteristic times with accuracy – even more when two spectra are used – yielding excellent viscoelastic predictions. It further requires relatively little parameter identification. It is however computationally expensive: To integrate the influence of so many characteristic times, the integral is discretized in a sum of several linear viscoelastic Kelvin-Voigt mechanisms (up to 200 in this study). With the addition of a nonlinear stress dependent function, it becomes necessary to solve a cumbersome Jacobian and to store of as many rank 2 tensors as elementary mechanisms. The principle of using a spectrum of characteristic times will be retained in our study and extended to the nonlinear regime, yet with a very different implementation.

Based on the state of the art, in the first section of this paper, a constitutive model taking into account the influence of strain rate in a simple and modular manner is presented. In the second section, the model parameters are identified thanks to rigorous characterisation and identification procedures, featuring an original dynamic strain interruption device. After a short discussion, ongoing work and perspectives conclude this paper.
2. MODEL CHARACTERISTICS

The model developed in this study is based on the modular approach proposed in [6]. For crash applications, only short term effects are added to the reference quasi-static model. Therefore the model features two main components:

- A nonlinear anisotropic damageable elastic-plastic model, thereafter referred to as the reference quasi-static model. No rate effects are considered other than a bounding of the damage rate for softening regularization purposes, which occur at much higher strain rates than those of interest in this study. That component of the model allows the reproduction of loss of stiffness and irreversible strains in all directions, up to failure, and is strongly inspired by other phenomenological approaches at the scale of the ply, such as [7] or [8].

- An spectral viscoelastic model based on a generalized Maxwell approach. The principle lies in integrating a Maxwell model over a continuous series of characteristic times, with a spectrum-shaped weight function. When a single transition rate is considered, the spectrum may be approximated by a bell function around that order of magnitude. But since we consider a wide range of strain rates, we prefer the use of two bell functions, or two spectra, in order to capture both a quasi-static and a dynamic transition rates, as in [4].

These two components are set up in parallel, i.e. the total strain is applied to both and their resulting stresses sum up to obtain the total apparent stress. Therefore the first and fundamental assumption of this approach is that of stress partition:

\[ \sigma = \sigma^{qs} + \sigma^{st} \]  

(2)

where \( \sigma^{qs} \) is the quasi-static stress component and \( \sigma^{st} \) is the short term stress component – or total viscoelastic overstress.

As previously argued, small strains are considered. The mechanical response is considered isothermal, at a temperature of 23 °C and a relative hygrometry of 50 %. Polyamide is very temperature and moisture sensitive [9] and extensions of the current model to other operating conditions requires a thorough analysis of environmental effects on the instantaneous response of such materials. We also assume strain partition in the quasi-static and short term branches:

\[ \varepsilon^{qs}_e + \varepsilon^{qs}_i = \varepsilon = \varepsilon^{st}_e(\tau) + \varepsilon^{st}_v(\tau) \]  

(3)

where \( \varepsilon^{qs}_e \) and \( \varepsilon^{qs}_i \) are respectively the elastic and irreversible strains in the quasi-static branch; and \( \varepsilon^{st}_e(\tau) \) and \( \varepsilon^{st}_v(\tau) \) are the elastic and viscous strains in the short term branch associated to characteristic time \( \tau \).

All contributions considered, the Helmholtz free energy density \( \psi \) and the dissipated power
density $\dot{\phi}$ of the material read:

$$\rho \dot{\psi} \left( \varepsilon^{qs}_e \varepsilon^{st}_e (\tau), \dot{d}, \dot{p} \right) = \frac{1}{2} \varepsilon^{qs}_e : \mathbb{H} (d) : \varepsilon^{qs}_e + \int_0^p R(q) \, dq$$

(4)

$$+ \frac{1}{2} \int_{\tau_-}^{\tau_+} \varepsilon^{st}_e (\tau) : \mu (\tau) \mathbb{V} : \varepsilon^{st}_e (\tau) \, d\tau$$

where all the parameters and variables introduced are defined in the following.

The quasi-static elastic strains feature a strong coupling with a damageable behavior, that diminishes the quasi-static elastic stiffness $H$ such that:

$$\sigma^{qs}_e = \rho \frac{\partial \psi}{\partial \varepsilon^{qs}_e} = \mathbb{H} (d_{i=1..k}) : \varepsilon^{qs}_e = \left( [8 \mathbb{I} - \sum_{i=1}^k 8 \mathbb{B}_i d_i] : \mathbb{H}^0 \right) : \varepsilon^{qs}_e$$

(6)

where $\mathbb{H}^0$ is the initial, undamaged Hooke matrix; $d_i$ is the $i$th component of a series of $k$ damage mechanisms; and $8 \mathbb{B}_i$ is the associated eighth order tensor, featuring multiaxial couplings and contributions ($8 \mathbb{I}$ is the eighth order identity tensor). Note that $8 \mathbb{B}_i$ is mostly empty for all mechanisms, so that $C_i = 8 \mathbb{B}_i : \mathbb{H}^0$ results in a conventional anisotropic quadratic operator between a few strain components.

From a thermodynamic point of view, the quasi-static stresses are the dual of the quasi-static elastic strains, and the forces $Y_i$ the duals of the damage variables $d_i$:

$$Y_i = -\rho \frac{\partial \psi}{\partial d_i} = \frac{1}{2} \varepsilon^{qs}_e : C_i : \varepsilon^{qs}_e \quad \forall \, i = 1..k$$

(7)

The damage rate is defined similarly for each variable:

$$\dot{d}_i = \frac{1}{t_c} \left( 1 - e^{-a (w_i (Y) - d_i)_+} \right)$$

(8)

where $t_c$ and $a$ are material parameters, $(\star)_+$ is the positive part function, and $w_i (Y)$ are the damage flow laws.

We adopt a normal flow rule for each damage law, although this is not mandatory and other authors [8] have shown that a non normal flow is in better agreement with the physics for certain mechanisms. For instance, if we consider diffuse fiber-matrix debonding and matrix microcracking as a single mechanism $d$, we have:

$$w_d (Y_d) = d^c_d \left( 1 - e^{-\sqrt{Y_d / Y_d^c}} \right)$$

(9)
where \( d_s^k \) is the maximum, saturating value of damage and \( Y_d^k \) affects damage kinetics.

If we consider tensile cracking of the warp tows, indexed \( f_{1} \):

\[
w_{f_{1}} (Y_{f_{1}}) = \begin{cases} 
0 & \forall \ Y_{f_{1}} < Y_{c}^{f_{1}} \\
1 & \text{otherwise} 
\end{cases} \tag{10}
\]

with \( Y_{c}^{f_{1}} \) the critical failure energy.

The quasi-static irreversible strains feature a nonlinear isotropic hardening, driven by effective stresses, so that the yield condition reads:

\[
f (\bar{\sigma}, p) = \sigma_{eq} (\bar{\sigma}) - R(p) - R_0
\tag{11}
\]

with:

\[
\sigma_{eq} = \sqrt{\bar{\sigma} : P : \bar{\sigma}} \tag{12}
\]

\[
\bar{\sigma} = \mathbb{H}^0 : \varepsilon_{qs}^e \tag{13}
\]

\[
R(p) = K p^m \tag{14}
\]

where \( R \) and \( p \) are respectively the hardening function and the effective plasticity; \( \sigma_{eq} \) and \( \bar{\sigma} \) are respectively the equivalent and effective stresses; \( P \) is a diagonal fourth order coupling tensor describing an anisotropic Hill yielding criterion; and \( K \) and \( m \) are hardening parameters.

The flow rule is chosen normal and associated. Strain equivalence is also assumed, so that:

\[
\dot{\varepsilon}_{qs}^e = \dot{p} \frac{\partial f}{\partial \sigma} = \dot{p} \frac{P : \bar{\sigma}}{\sigma_{eq}} = \dot{\varepsilon} - \dot{\varepsilon}_{qs}^e \tag{15}
\]

The short term stress results from the integration of a series of stresses, obeying to a series of linear Maxwell models, defined over a compact support \([\tau_-, \tau_+]\):

\[
\sigma^{st} = \int_{\tau_-}^{\tau_+} \chi (\tau) d\tau \tag{16}
\]

with:

\[
\chi (\tau) = \mu (\tau) \varepsilon^{st}_v (\tau) = \mu (\tau) \mathbb{N} : \varepsilon^{st}_v (\tau) \tag{17}
\]

where \( \mathbb{V} \) is a fourth order elastic tensor, that may be interpreted as the maximum extra stiffness provided by high speed loadings; \( \mathbb{N} \) is the dynamic viscosity tensor of the Newtonian fluid law; and \( \mu (\tau) \) is a weight function defined as:

\[
\mu (\tau) = \sum_i \left( \frac{\xi_i (\tau)}{\tau_+^{\xi_i (\tau)} \xi_i (\tau)} \right) \text{ where } \xi_i (\tau) = \frac{1}{\log (\tau^d_i \tau^d_i)} e^{-\left( \frac{\log (\tau^d_i) - \log (\tau^d_i)}{\sqrt{\pi}} \right)^2} \tag{18}
\]

with \( \tau^d_i \) the average characteristic time of the \( i \)th spectrum and \( \tau^d_i \) its standard deviation. None of the above equations are affected by quasi-static mechanisms and vice versa.

This model introduces a number of fourth order tensors and scalar parameters. Not all of them are identified on experimental results. The components of coupling tensors \( C_{i=1,k} \) and \( \mathbb{P} \), are obtained through homogenization considerations or by analogy with other studies [8, 10].

41
3. HIGH SPEED CHARACTERIZATION AND IDENTIFICATION PROCEDURES

The identification of visco-elastic parameters and their effect – or lack thereof – on damage and plasticity kinetics requires cycled loading sequences in order to discriminate the evolution of each internal variable. However dynamic tensile machines for intermediate speeds are usually servo-hydraulic machines. They feature an open loop loading system, where the hydraulic jack is thrown almost instantly at a prescribed speed, but cannot be stopped before the end of its course. Cycling the specimen therefore requires an interruption device to separate the sample from the hydraulic jack before total failure. A detailed account on the characteristics of a strain interruption testing device of our design may be found in [11]. We only remind its most relevant properties for the study at hand and discuss the results obtained on the material of interest.

The device, schematized in Figure 2, guides the movement of the bottom jaw with respect to the upper one using two steel columns. Tightened to the bottom jaw, these columns are flanged at their upper end, at a distance to the upper jaw that can be adjusted using mechanical arrests made of double nuts. When the hydraulic jack pulls the bottom jaw down, the testing sample is loaded instantly at a prescribed speed until the flanges come into contact with the mechanical arrests. Just before this happens, aluminum lamella slide into machined groove on the sides of the device, so that no elastic springback of the sample may occur afterwards. After contact is established, the sample remains loaded at the prescribed maximum displacement, which no longer increases. The load increases in the rest of the mechanism, until a brass fuse set up in series with the upper jaw fails and the device disconnects from the machine. The sample may then be relaxed and dismounted, a new fuse and new adjustable mechanical arrests set up, the sample reinstalled, and a new cycle may be applied.

Figure 2. Schematics of the strain interruption testing device
We use this device to submit testing samples of reduced dimensions – also discussed and validated in [11] – to seven strain increments each, at different prescribed strain rates\(^2\). For the 45° oriented samples, the unfiltered cycled responses we obtained for all four strain rates are shown in Figure 3. The 25 s\(^{-1}\) response suffers from excessive noise and is therefore discarded in the following. The rest of the curves offer a decent insight of the sequential loading behavior of the material. Two changes of curvature may be observed on some loading paths, likely due to finite strain non-linearities. As they appear only beyond 10% total strain and we lack unloading paths, we have decided not to focus our efforts in reproducing these tendencies. In addition, loss of apparent stiffness and significant residual strains may be observed.

\[ \sigma_{12} \text{ (MPa)} \]
\[ \dot{\varepsilon} = 5 \times 10^{-3} \text{s}^{-1} \]
\[ \sigma_{12} \text{ (MPa)} \]
\[ \dot{\varepsilon} = 5 \times 10^{-1} \text{s}^{-1} \]
\[ \sigma_{12} \text{ (MPa)} \]
\[ \dot{\varepsilon} = 2 \times 10^{1} \text{s}^{-1} \]

Figure 3. Cycled response of 45° oriented samples submitted to a variety of strain rates

The idea is now to properly and automatically identify all the in-plane model parameters using these results, preferably in a direct manner. Quasi-static cycled responses are also used for that purpose. A key point to our approach is that visco-elastic effects are entirely uncoupled with the rest of the model and may therefore be identified separately. Let us start with them, assuming that \( E_1 = E_2, \nu_{12}, \) and \( G_{12} \) have already been identified.

Six parameters are needed to take rate effects into account: The non zero components of

\(^2\)The strain rates are theoretical, as they rely on the measured speed of the bottom jaw normalized by the free length of the sample. The strain measurements are however performed using Digital Image Correlation (DIC) in a region of the sample where the strains are homogeneous, as a verification campaign confirmed.
the short term elastic tensor, $V_{1111} = V_{2222}$ and $V_{1212}$, and the characteristic times defining the average and standard deviation of the first and second spectra, $\tau_{a}^{I}, \tau_{d}^{I}, \tau_{a}^{II}$, and $\tau_{d}^{II}$ respectively. This part of the identification rests on a least square minimization between experimental and computed curves in the elastic regime, defined as $\sigma < \sigma_0 = 15$ MPa. For $n_c$ considered curves made of $n_v$ experimental values, the function to be minimized reads:

$$Q(p) = \frac{1}{n_c} \sum_{i=1}^{n_c} \frac{1}{n_v(i)} \sum_{j=1}^{n_v(i)} \left( \frac{\sigma_{cmp}(i,j,p) - \sigma_{exp}(i,j)}{\sigma_0} \right)^2$$

(19)

where $\sigma_{cmp}$ is the computed stress corresponding to a uniaxial load in the same direction as in the experiment, whose nominal stress is $\sigma_{exp}$. The vector $p$ may contain all six parameters or only a few, to perform a sequential identification. For instance, $V_{1111}$ is usually left out and identified after all other parameters, as rate effects are less pronounced in the fibers’ direction. After a first sensitivity analysis ensured the unicity of the solution and reasonable initial guesses for each parameter, a BFGS method is used to minimize Equation 19.

The identification of the nonlinear quasi-static parameters is based on the same principles as [10]. It however requires the extrapolation of straight lines for each cycle, in order to define the residual strain level and effective stiffness. In carbon-epoxy laminates, the thin, symmetric, simply curved hystereses observed during unloading/loading paths make for a simple procedure: The bisector may be used as the straight line. In our case where we lack all three of these properties, the procedure must be adapted. Further we also need to calibrate the initial value of each reloading path, as creep or recovery could not be measured in the samples after interrupted dynamic tension. Finally we first need to eliminate the visco-elastic contribution to each experimental response, since quasi-static mechanisms are considered independent.

The latter is taken care of by substracting the response of the now-identified visco-elastic component of the model to all experimental measurements. Then each loading path is divided

![Figure 4. Identification results of the visco-elastic parameters of the model in terms of monotonous response of 45° oriented samples (a) and tangent shear moduli (b)](image-url)
in three: The lower, concave part, is discarded, as discussed earlier; the middle, slightly convex part, is used to interpolate the slope of the line; finally the upper, concave part, corresponding to the active nonlinear response between two stress increments, is used to calibrate the abscissa of the line. With this automated procedure, we have fully defined the straight line figuring each loading cycle. The results are plotted in Figure 5 for three different strain rates. The monotonous responses differ slightly from the envelope of the calibrated cycled responses. Further the discarded lower parts of the loading paths are not captured at all. But except for these discrepancies, the trend of the nonlinear regime is well reproduced.

Figure 5. Monotonous (red dots) and cycled tensile response (blue) in 45° oriented samples; with the regions (green) used to extrapolate the straight lines (black lines)

With this procedure at hand, it is now trivial to identify the evolution laws of damage and irreversible strains independently. Unlike numerical integration, identification may be performed in any order, which leaves the choice of each law rather open and allows for much more refined models than we used here. After identification of their parameters, the laws described in Equations 14 and 9 are confronted to experimental measurements in Figure 6. Although hardening seems to be stiffer at higher strain rates, all curves tend to converge to a similar trend after a few percent strain. Damage kinetics appear even less rate sensitive.

Figure 6. Evolution laws of irreversible strains (a) and damage (b) plotted against experimental results at various strain rates
These results tend to confirm our initial idea of treating rate effects as extra stresses with no influence on the quasi-static nonlinear behavior. To assess exactly what is reproduced and what is lost compared to experimental values, a confrontation of the model with experimental curves is plotted in Figure 7 for two strain rates. Although a finer reproduction of some cycles for very small strains may be sought, the accuracy of the model, including on unloading/loading paths, is deemed more than sufficient. And with the addition of rate effects, the computed envelope fits the experimental one quite satisfactorily.

Figure 7. Confrontation of the model predictions with experimental responses on 45° oriented samples loaded at $5 \cdot 10^{-3}$ (a) and $5 \text{ s}^{-1}$ (b) strain rate

4. CONCLUSIONS AND PERSPECTIVES

In this paper we have presented a new constitutive model whose key characteristic is its modularity, which allows its tailoring for specific materials and loading conditions. It was applied to a woven glass fiber polyamide matrix composite, submitted to high speed loads. To capture the significant strain rate effects that were observed, a spectral visco-elastic model based on a generalized Maxwell approach was used. No couplings between visco-elasticity and other nonlinearities were introduced, which eases both numerical integration and identification. A dedicated characterization procedure, involving a recent strain interruption testing device designed to allow dynamic cycled loading, was also proposed to assess the influence of strain rate on evolution laws. The parameters of the model were identified based on the direct evaluation of quasi-static quantities and on the indirect calibration of visco-elastic ones. It confirmed that no further strain rate dependency is needed for irreversible strains nor loss of stiffness. The model reproduces satisfactorily both the envelope curves and the cycled loading paths, despite the fact that rate effects were only introduced in the spectral visco-elastic model.

Current investigations include the application of such a model to structure computations, in which multiaxial loads and gradients of strain rate may appear. The failure locus and predicted damage value are also of critical importance for the design of automotive structures submitted
to high energy impact. The identification of the bounded rate parameters $t_c$ and $a$, which was not discussed here, is one of the issues to be addressed.

References


A CYCLIC COHESIVE ZONE MODEL FOR VARIABLE AMPLITUDE LOADING AND MIXED–MODE BEHAVIOR

Martin Springer\textsuperscript{1,2,*}, Albert Turon\textsuperscript{2}, Heinz E. Pettermann\textsuperscript{1}

\textsuperscript{1} Institute of Lightweight Design and Structural Biomechanics, Vienna University of Technology
\textsuperscript{2} AMADE, Polytechnic School, University of Girona, Spain
\textsuperscript{*} springer@ilsb.tuwien.ac.at

Key words: Cohesive Zone Model, Fatigue, Non-Local, Variable Amplitude Loading.

Summary: A Cyclic Cohesive Zone Model is developed which is an extension of an exponential traction–separation law describing the quasi–static constitutive behavior of the interface. Damage degradation under fatigue loading is accounted in a cycle–by–cycle analysis. A Paris’ law type formulation is utilized to model fatigue damage which is based on physically interpretable interface properties, overcoming the need of any parameter fitting. Mixed–Mode loading is considered by the BK–criterion. Varying load amplitudes are captured by the model formulation. For constant amplitude loading or slowly changing load amplitudes a cycle jump technique is implemented to decrease the computational time. The model is implemented within the framework of the Finite Element Method and includes non-local evaluation of structural parameters. Finally, the proposed model is demonstrated on Mode I, Mode II, and Mixed–Mode delamination tests of carbon/epoxy laminates.

1. INTRODUCTION

Cyclic loading of layered composite structures may lead to various types of failure mechanisms inside the laminate. One of these failure mechanisms is fatigue driven delamination between the plies. Such delaminations compromise the structural integrity of a component and lead to a loss in the load carrying capability.

In terms of the Finite Element Method (FEM), traction–separation laws are widely used to model delamination growth of interfaces. Different models have been developed to account for fatigue driven delamination. Cyclic Cohesive Zone Models (CCZM) can be categorized into hysteresis loop damage models and envelope load damage models \cite{1}. Hysteresis loop damage models are based on a loading–unloading formulation, which enables the consideration of varying load amplitudes during a fatigue analysis, Fig. 1(a). Damage development at the interface is considered by interfacial stiffness and strength reduction. The reduction is described by damage evolution laws which are often based on fitting parameters. Calibration of such models can be a difficult task and the predictive capabilities of such formulations are limited. In contrast, for higher numbers of load cycles, so called envelope load damage models are utilized, Fig. 1(b). They are based on a Paris’ law like formulation and rely on physically interpretable parameters. Additionally, these models can consider effects such as mixed-mode, loading ratio,
and delamination thresholds. These advantages in the predictive capabilities of envelope load damage models compared to hysteresis loop models are most often limited by formulations considering only blocks of constant load amplitudes.

The proposed CCZM in this work utilizes a hysteresis loop damage formulation to account for variable amplitude loading in combination with a fatigue damage evolution law based on interface properties without parameter fitting. The developed CCZM is based on the quasi–static formulation from [2], which utilizes an exponential traction–separation law to describe the interface behavior. Fatigue damage is accounted for by a Paris’ law like formulation based on [3], where the fatigue damage rate formulation includes structural parameters which are approximated at the material level. In contrast, the proposed model in this work circumvents the need for parameter approximation by utilizing a non–local evaluation of the structural properties directly during the simulation. The non–local constitutive law is implemented into ANSYS® Mechanical APDL by means of USERINTER user contact subroutine.

2. CONSTITUTIVE MODEL

The proposed Cyclic Cohesive Zone Model (CCZM) is derived in this section. The CCZM is based on the quasi–static formulation of [2] and utilizes an exponential traction–separation law. Mixed–mode behavior is accounted for by the BK–criterion. This formulation is reviewed and summarized for single–mode and mixed–mode failure. The quasi–static model is extended by a fatigue damage rate formulation based on [3], where a Paris’ law like approach describes the evolution of the damage variable during fatigue loading. A non–local evaluation is introduced to address the structural parameters within this formulation. A simple cycle jump
technique mostly for constant amplitude loading is employed to reduce the computational time. The CCZM is implemented within the framework of the Finite Element Method (FEM) and important implementation aspects are provided next to the model development. Finally, the actual model response is illustrated by analyzing a structural problem undergoing subcritical crack growth.

2.1 SINGLE–MODE DELAMINATION GROWTH FOR QUASI–STATIC LOADING

The CCZM builds up on an irreversible constitutive law for steady–state crack growth [2]. By considering only single–mode failure the exponential traction–separation law gives the interfacial traction, as,

\[ T(\Delta) = T^0 \Delta e^{\frac{1-\beta}{\beta}} \quad \text{with} \quad \Delta = \frac{\Delta}{\Delta^0}. \]

(1)

\( \Delta \) is the normalized separation. \( T^0 \) and \( \Delta^0 \) define the point of damage onset. The traction–separation law is depicted in Fig. 2(a). Interface failure is accounted for by correlating the area below the traction–separation law to the critical energy release rate as,

\[ G_c = \int_0^{\infty} T(\Delta) d\Delta = T^0 \Delta^0 \psi(\beta) \quad \text{with} \quad \psi(\beta) = \beta^{2-\beta} \Gamma(\frac{2}{\beta}) e^{\frac{1}{\beta}}. \]

(2)

Herein \( \beta \in \mathbb{R}^+ \) is a parameter influencing the shape of the cohesive law and \( \Gamma(\bullet) \) is the Euler gamma function. The traction–separation law as defined in Eq. (1) holds only true for monotonically increasing loading. The work caused by opening of the interface is still recoverable at closing. Hence, a damage formulation is included and damage progression is accounted for by an elasto–damage model. An internal state variable is introduced and a nonlinear unloading path back to the origin results from the formulation. The modified traction–separation law reads [2],

\[ T(\Delta, d_s) = T^0 \Delta e^{\frac{2-\beta}{\beta}} \frac{\Delta^\beta}{d_s - d_s}, \]

(3)

where \( d_s \) is the internal state variable describing the damage state. Since thermodynamics requires irreversible energy dissipation associated with a damage process, the internal damage variable needs to fulfill the following restrictions

\[ d_s = \begin{cases} \Delta^\beta & \text{if} \quad \Delta^\beta \geq d_s \\ 0 & \text{if} \quad \Delta^\beta < d_s \end{cases}. \]

(4)

In a more convenient form the evolution of \( d_s \) can be formulated as,

\[ t+\delta t d_s = \max(1, t d_s, t+\delta t \Delta^\beta), \quad 0 d_s = 1, \]

(5)

where \( t \) is the time and \( \delta t \) the time increment. The internal state variable ranges from 1 to \( \infty \). This is caused by the exponential nature of the cohesive law. In the framework of continuum
Figure 2: (a) Traction–separation law of the quasi–static formulation, (b) Traction–separation law accounting for fatigue damage. Shading indicates the amount of the dissipated energy rates during the damage process caused by quasi–static, $W_s$, and fatigue, $W_f$, loading.

damage mechanics, damage variables in the range from 0 to 1 are widely used. Hence, an energy definition is introduced for the quasi–static damage variable,

$$D_s(\Delta_{\text{max}}) = \frac{W_s}{G_c} = \frac{1}{G_c} \left[ \int_0^{\Delta_{\text{max}}} T(\Delta) \, d\Delta - \int_0^{\Delta_{\text{max}}} T(\Delta, d_s) \, d\Delta \right]$$

which is defined by normalizing the energy dissipation rate, $W_s$, with the critical energy release rate, $G_c$. $W_s$ is computed by calculating the area under the undamaged constitutive law, $T(\Delta)$, up to the maximum separation, $\Delta_{\text{max}}$, and subtracting the area below the damaged constitutive law, $T(\Delta, d_s)$, up to the same separation, $\Delta_{\text{max}}$, see Fig. 2(a). Thus, $D_s = 0$ describes the undamaged interface and $D_s = 1$ the completely damaged interface.

2.2 MIXED–MODE DELAMINATION GROWTH FOR QUASI–STATIC LOADING

Mixed–mode loading is accounted defining an interaction criterion to obtain an equivalent mixed–mode fracture toughness. In this work the BK-Criterion is used. The quasi–static formulation is already derived for 3D in [2]. The fatigue damage extension is currently limited to 2D, considering only Mode I and Mode II crack growth. Hence, only the necessary equations for 2D of the quasi–static constitutive law are summarized below. The interfacial constitutive equations are obtained as,

$$\begin{bmatrix} T_1 \\ T_3 \end{bmatrix} = \begin{bmatrix} \frac{\Delta_1}{\langle \Delta_3 \rangle} \end{bmatrix} S e^{-\frac{\mu^0/d_s - d_s}{\eta}},$$

where 1 denotes the shear direction and 3 the normal direction of the interface. The Macaulay brackets are defined as $\langle x \rangle = \max(0, x)$ and $S = 1 + (\nu - 1)B^\eta$ where $\nu = G_{Ic}/G_{IIc}$ and $\eta$ is
the BK-exponent.

\[ B = \left( 1 - \frac{\langle \Delta_3 \rangle^2}{\lambda^2} \right) \]  

(8)

is the so called mode mixity. The effective separation is given by,

\[ \lambda = (|\Delta_1|^2 + \langle \Delta_3 \rangle^2)^{\frac{1}{2}} \]  

(9)

and the evolution of the internal state variable follows

\[ t^{+\delta t} d_s = \max(1, t_s, t^{+\delta t} \lambda^2) \]  

(10)

The growth criterion is given as,

\[ G_c = G_{Ic} + (G_{IIc} - G_{Ic}) B \eta, \]  

(11)

where \( G_{Ic} \) are the critical energy release rates for Mode I and Mode II, respectively. The onset criterion for mixed-mode loading,

\[ (T_o^o)^2 = (T_o^3)^2 + \left( (T_o^1)^2 - (T_o^3)^2 \right) B \eta, \]  

(12)

is based on a BK-like criterion [4]. Additionally, under Mode I loading penetration of the interface surfaces must be avoided. Therefore, a contact algorithm needs to be employed. A penalty contact formulation is utilized to compute the pressure–penetration relation,

\[ T_3 = K \Delta_3, \quad \text{for} \quad \Delta_3 < 0 \]  

(13)

where \( K \) is the contact stiffness.

### 2.3 EXTENSION OF THE DELAMINATION MODEL FOR FATIGUE LOADING

Fatigue damage cannot be accounted for by the quasi–static formulation. This is seen in Fig. 2(a) where the unloading–reloading path remains the same. Hence, no damage is accumulated during cyclic loading and the stiffness and strength of the interface remains unchanged as developed during the previous loading up to \( D_{\text{max}} \). To account for fatigue damage the model is extended by introducing a total damage rate,

\[ \frac{dD_{\text{tot}}}{dN} = \frac{dD_s}{dN} + \frac{dD_f}{dN}, \]  

(14)

with the number of load cycles, \( N \). The total damage rate is split up into a part related to quasi–static damage, \( \frac{dD_s}{dN} \), and a part that is related to fatigue damage, \( \frac{dD_f}{dN} \). The first term is obtained by the quasi–static damage model. For the evolution of the second term, the fatigue damage rate, different formulations have been proposed. A review can be found in [5]. The proposed model utilizes the fatigue damage rate formulation of [3],

\[ \frac{dD_f}{dN} = \frac{1}{l_{cg}} P(\Delta w) \]  

(15)
Herein, $l_{cz}$ is the length of the cohesive zone and $P(\Delta w)$ is a Paris’ law like function dependent on the specific work range, $\Delta w$, of each interface point in the cohesive interface. This Paris’ law like function defines a local delamination growth rate and is determined locally. The length of the cohesive zone is a structural property and depends on the geometry of the structure, the loading conditions and the mixed–mode. In the original formulation [3] it is approximated. In the present formulation the approximation is circumvented by determining $l_{cz}$ with a non-local approach during the FEM simulation. In the context of the FEM $l_{cz}$ is determined at the end of each load cycle and defined by,

$$l_{cz} = \sum_e l_e \quad \text{for} \quad \{e \mid 0 < D_e < 1\} \quad , \quad (16)$$

where $l_e$ is the element length of the cohesive element, $e$. This way, the interface length is measured from the crack tip to the undamaged interface. The length of the cohesive zone is therefore a direct result of the simulation and no assumptions have to be made. The Paris’ law like function in the fatigue damage rate formulation reads,

$$P(\Delta w) = \begin{cases} C \left( \frac{\Delta w}{G_c} \right)^m & \text{for} \quad \Delta G_{th} < \Delta w < G_c \\ 0 & \text{for} \quad \Delta w \leq \Delta G_{th} \quad , \quad (17) \end{cases}$$

where the range of the specific work for each interface point is computed by

$$\Delta w = w_{max} - w_{min} = \int_{\Delta_{min}}^{\Delta_{max}} T(\Delta) d\Delta \quad . \quad (18)$$

The determination of $\Delta w$ is depicted in Fig. (3) and the ratio between the maximum and minimum specific work can be interpreted as the R-ratio, $R = w_{min}/w_{max}$. $G_c$ is the fracture toughness and $G_{th}$, $C$, and $m$ are the fatigue parameters obtained from experimental results. The most straightforward approach to obtain these properties is to assume that the local development of the fatigue damage is comparable to the macroscopic crack growth rate. Then the parameters result directly from experimental $da/dN$ curves. The typical pattern of a crack growth rate is divided into three regions. A region where no crack growth is observed, which means the energy release rate, $G$, is lower than a threshold value, $G_{th}$. Contrary, when $G$ reaches the fracture toughness, $G_c$, steady–state crack growth is obtained. Between this two regions a Paris’ type law is taken,

$$\frac{da}{dN} = \begin{cases} C \left( \frac{\Delta G}{G_c} \right)^m & \text{for} \quad \Delta G_{th} < \Delta G < G_c \\ 0 & \text{for} \quad \Delta G \leq \Delta G_{th} \quad , \quad (19) \end{cases}$$

where the parameters $C$ and $m$ are structural properties determined by experiments. For a general mixed–mode load case, the coefficients $C$ and $m$ of the Paris’ law are calculated by the
Figure 3: Sketches of the specific work range, $\Delta w$, at an interface point computed by Eq. (18).

The following formulae [6],

$$\log(C) = \log(C_1) + \phi \log(C_m) + \phi^2 \log \left( \frac{C_{II}}{C_m C_1} \right)$$

$$m = m_1 + m_m \phi + (m_{II} - m_1 - m_m) \phi^2,$$

where $C_1, C_{II}, m_1,$ and $m_{II}$ are the pure mode parameters and $C_m, m_m$ are fitting parameters, all obtained by experiments. The mode ratio, $\phi = \frac{G_{II}}{G}$, is a structural property, where $G = G_1 + G_{II}$ is the total energy release rate computed by the energy release rates for Mode I, $G_1$, and Mode II, $G_{II}$. The fatigue threshold value is defined in a similar way as the growth criterion, as,

$$G_{th} = G_{Ith} + (G_{Ith} - G_{Ith}) \phi^n.$$

Variable amplitude loading can lead to a change of the mode ratio during a load cycle. But experimental crack growth curves are usually available for loading conditions with fixed mode ratios only. Therefore, a fracture criterion is defined to evaluate $\phi$ at the time,

$$t_{BK}^* = \arg \max_{t \in [t_n, t_{n+1}]} \left\{ \frac{G(t)}{G_{Ic} + (G_{Ic} - G_{Ic}) \phi^n(t)} \right\}.$$

The criterion relates the energy release rate, $G(t)$, during the load cycle to a critical energy release rate predicted by a modified BK–criterion. The maximum of this criterion is assumed to represent the time at the critical loading conditions, i.e. the highest contribution to the damage development. In Linear Elastic Fracture Mechanics the J-Integral equals $G$ and the mode decomposition of the J-Integral [7] results in,

$$J = J_1 + J_{II} = \mathcal{G}_1 + \mathcal{G}_{II} = \left( -\int_0^{l_2} T_3 \frac{\partial \Delta_3}{\partial x} \, dx \right) + \left( -\int_0^{l_2} T_1 \frac{\partial \Delta_1}{\partial x} \, dx \right).$$
the structural properties $G_I$ and $G_{II}$. In terms of the FEM these parameters are evaluated as,

$$G_I(t_{BK}) = \sum_{e \in \mathcal{E}_{lc}} \left[ \left\langle T^e_3 \right\rangle \left| \frac{\partial \Delta^e_3}{\partial x} \right|_{e} \right] l_e = \sum_{e \in \mathcal{E}_{lc}} \left[ \left\langle T^e_3 \right\rangle \left| \Delta^b_3 - \Delta^a_3 \right|_{e} \right]$$  (25)

$$G_{II}(t_{BK}) = \sum_{e \in \mathcal{E}_{lc}} \left[ T^e_i \left| \frac{\partial \Delta^e_i}{\partial x} \right|_{e} \right] l_e = \sum_{e \in \mathcal{E}_{lc}} \left| T^e_i \left( \Delta^b_i - \Delta^a_i \right) \right|_{e}.$$  (26)

The derivatives, $\partial \Delta_i / \partial x$, are computed at the element level. This is done in a discrete way for cohesive elements with linear interpolation functions as $\partial \Delta_i / \partial x = (\Delta^b_i - \Delta^a_i) / l_e$, where $a$ and $b$ denote the nodes of the element and $l_e$ the element length.

With the quasi–static damage evolution and the fatigue damage rate, the total damage variable is obtained as,

$$(N_n+1) D_{tot} = (N_n+1) D_s + (N_n) D_f + \left. \frac{dD_f}{dN} \right|_{(N_n+1)} ,$$  (27)

where $(N_n+1)$ denotes the last simulated load cycle. The static damage variable, $D_s(\Delta_{max})$, depends only on the maximum separation arising during the whole loading history and is therefore known at the end of cycle $(N_n+1)$. The fatigue damage rate is computed for cycle $(N_n+1)$ according to Eq. (15) and added to the accumulated fatigue damage, $D_f$, until cycle $(N_n)$. Similar to the definition of the quasi–static damage variable, Eq. (6), the total damage variable is defined as,

$$D_{tot} = \frac{W_d}{G_c} = \frac{W_s + W_f}{G_c} ,$$  (28)

where $W_d$ is the total rate of the energy dissipated during the damage process. $W_s$ and $W_f$ describe the rates of the energies dissipated due to quasi–static and fatigue loading, respectively. The cyclic traction–separation law indicating the different energy rates as depicted in Fig. 2(b). The energy based definition of $D_{tot}$ allows for the update of the internal state variable, $d_{tot}$. This two variables are related in a nonlinear manner,

$$D_{tot}(d_{tot}) = \frac{1}{G_c} \left[ \int_0^{\Delta_{tot}} T(\Delta) \, d\Delta - \int_0^{\Delta_{tot}} T(\Delta, d_{tot}) \, d\Delta \right] ,$$  (29)

where $d_{tot} = \Delta_{tot}^\beta$ and $\Delta_{tot}$ can be interpreted as a fictitious separation which would be necessary to produce the same amount of damage as caused by both the quasi–static and fatigue loading. $\Delta_{tot}$ in Eq. 29 cannot be expressed explicitly and an iterative solution scheme is necessary. A Newton–Raphson solver is employed to determine $\Delta_{tot}$ at the end of each simulated load cycle and finally $d_{tot}$ is updated.
2.4 CYCLE JUMP STRATEGY

A cycle by cycle analysis can get computationally expensive for a high number of load cycles. The computational time is reduced by implementing a simple cycle jump technique. The cycle jump approach is limited to constant load amplitudes or slowly changing load amplitudes. The present strategy is focused on load spectra which can be represented by different load blocks of constant load amplitudes. Within this load blocks the cycle jump technique reduces the computational time but captures all the transient effects from higher to lower load amplitudes or vice versa. The total damage variable after the cycle jump reads,

\[(N_n + \Delta N) D_{tot} = (N_n) D_{tot} + \int_{N_n}^{N_n + \Delta N} \frac{dD_{tot}}{dN} dN,\]

where \((N_n)\) is the last load cycle for which the total damage variable has been evaluated. \(\Delta N\) is the increment in cycles for the cycle jump approach. The damage rate, \(dD_{tot}/dN\), is not known for \(N \in ([N_n + 1), (N_n + \Delta N)]\) and therefore the integral on the right hand side of the equation cannot be determined exactly. In the presented model a 1–point Newton–Cotes quadrature is utilized and the total damage variable is approximated as,

\[(N_n + \Delta N) D_{tot} \approx (N_n) D_{tot} + \Delta D_s + \Delta N \frac{dD_f}{dN} \bigg|_{(N_n+1)},\]

where \(\Delta D_s \approx (N_n+1) D_s - (N_n) D_s\). To keep the error caused by this approximation within reasonable bounds, the increment in cycles of the cycle jump technique is limited by,

\[\Delta N = \frac{\Delta D_f^{allow}}{\max_i \{\left(\frac{dD_f}{dN}\right)_i\}},\]

where \(\Delta D_f^{allow}\) is the maximum allowable damage increment for one cycle jump and is divided by the highest fatigue damage rate in the cycle \((N_n + 1\), usually occurring at the crack tip.

2.5 ACTUAL MODEL RESPONSE

The actual model response of the CCZM is presented. The complete response of the model can only be demonstrated by considering a structural analysis undergoing subcritical crack growth (e.g. a specimen experiencing cyclic interface crack growth). The traction–separation response is monitored of one fixed point within the interface. This way, the whole delamination process experienced by this point is captured and ranges from local damage onset to local delamination. The subcritical damage progress is depicted by the traction–separation response of the interface point, Fig. 4. Three different structural loading conditions are applied corresponding to normalized energy release rates of \(\Delta G/G_c = [0.3, 0.5, 0.8]\) with an R-ratio of \(R = 0.1\).
Figure 4: Traction–separation response of a fixed interface point experiencing fatigue delamination growth. The whole degradation process is depicted ranging from local damage onset to local interface failure. Three different structural loading conditions are applied to illustrate the influence of the increasing interface opening caused by delamination growth onto the local degradation process.

The nominal energy release rates are indicated by the grey areas below the quasi–static traction–separation response. Beginning at the lowest load level, $\Delta G/G_c = 0.3$, the maximum separation at the material point is nearly constant during the whole degradation process. Therefore, the separation at damage onset and the separation shortly before crack growth are roughly the same. Hence, there is only a neglectable change in the damage predicted by quasi–static loading under this loading conditions. Almost all of the energy dissipated during the degradation process was predicted by the fatigue damage rate. In contrast, for higher loading amplitudes the influence of the quasi–static damage rate rises. The maximum separation from damage onset until crack growth increases and consequently leads to an increase of the quasi–static damage variable. This accumulation of quasi–static damage during the opening of the interface leads to stiffness and strength degradation. Unloading back to the origin occurs then on a different path at decreased interface tractions. This is seen by the small hysteresis loops recognizable in the last figure at a load level of $\Delta G/G_c = 0.8$.

3. APPLICATION

Simulations of Mixed Mode Bending (MMB) tests are utilized for the demonstration of the Cyclic Cohesive Zone Model. The geometry of the specimen and the test setup, the loading conditions, as well as the carbon/epoxy laminate’s elastic, interface, and fatigue properties are taken from [7]. The MMB specimens are subjected to fatigue loading with constant load amplitudes. Various simulations with different mode ratios of $\phi = [0, 0.5, 1]$ and different load amplitudes are conducted. The bending moments for pure Mode I and pure Mode II simulations are chosen in such a way that the normalized energy release rate results in $\Delta G/G_c = [0.3, 0.5, 0.8, 0.9, 0.95]$ with an R-ratio of $R = 0.1$. Mixed–mode loading simulations are conducted at the following load levels $\Delta G/G_c = [0.3, 0.4, 0.6, 0.8, 0.9, 0.95]$ considering the same R-ratio as before. The bending moment loading of a MMB test ensures a constant energy release rate during crack propagation. The fatigue delamination propagation rate during the tests (and simulations) re-
mains therefore constant and simplifies the evaluation of the crack propagation rate. The simulations were stopped after the fatigue delamination has grown for a predefined length. This crack length is then divided by the number of load cycles obtained from the simulation and provides the average crack propagation rate, depicted in Fig. 5. The numerical results are compared to the experimental data from [8]. Additionally, the Paris’ laws obtained from the experimental data and utilized as input for the simulations are depicted as well. The numerical results fit closely to the Paris’ laws for Mode I and Mode II. For mixed–mode loading the crack propagation rates are somewhat underestimated for high numbers of load cycles, probably caused by the cycle jump technique, but show again good agreement for a smaller number of load cycles.

4. CONCLUSIONS

A Cyclic Cohesive Zone Model (CCZM) based on an exponential traction–separation law has been formulated together with a BK-Criterion for mixed–mode loading. The quasi–static formulation has been extended using a Paris’ law type formulation for the prediction of fatigue damage. The fatigue damage evolution law utilizes three structural parameters, the critical energy release rate, the cohesive zone length and the mode ratio. The latter two parameters are evaluated by a non–local approach during the simulation. This way, the CCZM formulation relies only on physically interpretable interface parameters and parameter fitting is not necessary. The total damage variable is defined as the ratio between the dissipated energy rate during the damage process and the critical energy release rate describing the damage state of the cohesive interface. The model has been demonstrated on simulations of Mixed Mode Bending tests and the predicted crack propagation rates are in agreement with the experimental data.
ACKNOWLEDGMENTS

This work was carried out during a research stay of M.S. at the AMADE institute at the University of Girona, Spain. The funding of M.S. by the Austrian Research Promotion Agency (FFG, Project No. 846579), the Carinthian Economic Promotion Fund (KWF-1521 l 26876 l 38867) and the KUWI grant from the TU Wien are greatly acknowledged.

References


EXPERIMENTAL INVESTIGATION OF A NOVEL HYBRID INTERLOCKING ALUMINIUM/COMPOSITE JOINT AT THE SINGLE FEATURE SCALE


*School of Engineering,
University of Limerick, Limerick, Ireland
Mark.Obrien@ul.ie

Keywords: Hybrid, Single lap shear, Adhesively bonded joint.

Summary: A hybrid joining technology between aluminium and carbon fibre reinforced plastics (CFRP) is presented for lightweight structural applications. Raised rectangular features were machined into the aluminium adherends with a corresponding depression machined into the CFRP adherend. The two materials were joined through secondary bonding with the feature and depression interlocked. This new joining technology was investigated using miniature single lap shear joints with a single interlocking feature allowing examination of the failure process under both scanning electron microscopy (SEM) and Micro CT scanning. Comparisons with simply bonded joints have shown significant improvements in shear strength and work to failure due to altered failure mechanisms.

1. INTRODUCTION

A key driver in the automotive industry over the last two decades is the need for more lightweight and efficient vehicles. As part of this drive, a significant amount of research has been conducted in the field of lightweight materials leading to a surge in the use of lightweight aluminium alloys and fibre reinforced plastics (FRP) [1, 2]. Due to high cost and concerns around recyclability and incompatibility with standard automotive manufacturing methods, FRPs are still not ubiquitous in the automotive sector outside of high end vehicles. The use of bi-material aluminium/carbon fibre reinforced (CFRP) structures has been investigated to minimise these materials’ drawbacks without sacrificing their desirable mechanical properties [3]. This form of aluminium/CFRP structure delivers the desired strength and stiffness at half the weight when compared to a full aluminium structure, while also costing less than a full CFRP structure [4].

As reductions in weight have become more difficult to achieve, increased attention has turned on creating more lightweight and efficient joints. The use of mechanical fixation with bolts and rivets have been used extensively in the automotive and aerospace industries [5], prompting a significant amount of research into their mechanical behaviour [6]. The simplicity of application combined with high strength and reliability have made these mechanical joining methods the industry standard. However, these methods create a significant increase in weight while also creating damage in the CFRP adherend leading to highly inefficient joints [7]. Adhesive joining
creates a far more efficient bond leading to a stiffer and stronger joint while also creating a barrier between dissimilar materials, reducing the risk of corrosion [8]. However, adhesive bonds are highly sensitive to the quality of the adherend surfaces and tend to have large deviation in ultimate failure strength [9]. These flaws require adhesive joints to be overengineered to ensure premature failure does not occur.

The shortcomings in traditional joining methods have led to the development of a number of hybrid joints which utilise a combination of adhesive and mechanical fastening in order to maximise the potential of both technologies. Initially hybrid approaches used traditional mechanical fastening methods such as bolts and rivets in conjunction with adhesive bonding to create redundant failure paths [10]. Although this method prevents catastrophic failure, there is still a substantial amount of damage created in the CFRP along with significant weight gains [11]. More recently, several alternate technologies have been proposed in an effort to minimise damage in the CFRP adherends. The majority of new hybrid joining technologies are based on hybrid bonded/pinned geometry where protrusions are additively manufactured onto the metallic surface. The FRP is pressed over these protrusions before co-curing the adherends, restricting fibre damage [12]. As the technology has improved, more complex pin shapes have been proposed to deliver further increases in strength [13, 14]. Pin geometry with undercuts such as ball headed pins display the greatest strength and create a perfectly plastic region before ultimate failure. However, the undercut causes the joint to fail via pin failure leading to catastrophic failure whereas straight pins are gradually pulled out leading to a progressive failure mechanism [15].

In this work, a novel fastenerless composite to metal interlocking joint geometry is proposed with the goal of increasing the strength and work to failure over that of a standard flat adhesive joint. An investigation into the effect of the interlocking geometry with different lengths and widths on the strength, work to failure and failure type of a single lap shear joint loaded under tension is presented. The effects of a single interlocking feature on microtensile specimens were investigated in order to study the effects of an individual feature.

2. EXPERIMENTAL

2.1 Materials

The single lap shear joint adherends were constructed of AA5754 aluminium, an alloy commonly used for inner structural parts in the automotive industry, and HTA/6376, an aerospace grade CFRP material whose properties are given in Table 1. In order to examine the hybrid interlocking joint at a single feature scale miniaturised single lap joints were constructed with dimensions as shown in Figure 1a. The CFRP adherends were manufactured with a layup of [0°/0°/90°/90°]2s using a vacuum assisted autoclave process at 175°C and 700 kN/m² resulting in a laminate with a nominal thickness of 2mm. The adherends were assembled using a mould to ensure the correct overlap was maintained across samples. Henkel Hysol 9466, a high strength epoxy with a room temperature cure cycle, was used to bond the adherends. The nominal bond thickness of 55µm was obtained through the use of glass microspheres embedded in the adhesive [16].
2.2 Single Lap Shear Testing

The single lap shear (SLJ) testing was carried out in a Deben Microtensile stage as shown in Figure 1b. The microtester was small enough to fit inside the vacuum chamber of a Jeol JSM 5600 scanning electron microscope (SEM) allowing in situ observation of the failure mechanisms during testing. The dimensions of the microtensile specimens are shown in Figure 1a where the restricted travel of the apparatus limited the length of the joints to a maximum of 44 mm. In addition, the CFRP adherends were scanned in a Micro CT scanner to assess the amount of internal damage present after joint failure.

2.3 Geometry

Three designs (C1, C2 and C3) were selected to investigate the effect of length and width of the interlocking feature on joint performance with configuration 0 (C0) acting as reference standard adhesively bonded joint. Configuration 1 (C1) is a rectangular profile with its major dimension in the loading direction while configuration 2 (C2) is a rectangular profile with the major dimension transverse to the loading direction. Configuration 3 (C3) is a square profile centred in the bondline with a small length and width. Configuration 4 (C4) is a plane strain design which was used for direct observation of crack growth and damage progression immediately adjacent to the feature under a scanning electron microscope (SEM). All designs and dimensions are shown in Figure 2. The aluminium adherends were machined with the male interlocking features using a high speed milling process while the CFRP adherends were machined using a laser machining process.
2.4 Surface Preparation

The aluminium samples’ surfaces were prepared using a sodium hydroxide etching solution. The surfaces were first cleaned with an acetone wipe, placed in 0.5M sodium hydroxide solution and immersed in an ultrasonic bath for 30 minutes after which they were rinsed with water for five minutes. The samples were then placed in an oven at 80°C to dry fully. The CFRP bond surfaces were initially cleaned with an acetone wipe followed by disk sanding the flat sections using P320 silicon carbide paper. The laser machined features were then grit blasted with 220 grit aluminium oxide. The grit blasting process used a 1mm nozzle at a pressure of 4 bar held 50mm from the surface. Compressed air was used to remove any remaining aluminium oxide before the bonded surfaces were cleaned with acetone a second time [17, 18]. To obtain an adequate surface for viewing under the SEM, the edges of the bonded samples were sanded with successively finer silicon carbide paper, from P240 to P2500. Subsequently, 0.3µm and 0.05µm alumina suspensions were used to bring the surface to a high polish. The samples were then coated with a 10nm gold coat to avoid charging in the SEM.

3. RESULTS & DISCUSSION

3.1 Joint Shear Strength

The average maximum shear strengths for joints C0, C1, C2 and C3 as well as their corresponding standard deviations are displayed in Figure 3. All three interlocking configurations delivered improvements with joints C1, C2 and C3 giving improvements of 18%, 10% and 8%, respectively, as compared to the standard, flat C0 joint. These improvements are delivered due to the interlocking feature preventing relative adherend displacement. In a standard flat lap joint, the adhesive alone prevents the adherends from displacing relative to one another. Failure of the joint occurs when the relative displacement of the adherends brings the adhesive to its failure
strain creating cracks in the adhesive which propagate through the bondline [19]. However, the presence of the interlocking features create an additional impedance to relative adherend displacement and prevents the adhesive reaching its failure strain without plastic deformation of the adherends. The size and shape of the profile therefore influences the ultimate shear strength of the joint, whereas the flat joint relies solely on the adhesive strength and bond quality. As the adhesive begins to yield, the interlocking features transfer load into the adherends preventing relative adherend displacement. This load transfer creates two possible avenues of failure:

1. **Adhesive Failure** whereby the load transfer through the interlocked feature causes shearing of the aluminium feature, allowing the adhesive bondline to reach its failure strain. The entirety of the bondline fails with little to no damage present in the CFRP adherend as illustrated in Figure 4a.

2. **Delamination Failure** whereby the load transfer causes crack growth in plies situated between the interlocking feature and the free end of the CFRP adherend. These cracks grow through the ply until reaching a ply boundary at which point delamination occurs leading to shearing out of the adherend section between the free end of the CFRP adherend and the interlocking feature. The adhesive bondline does not completely fail with the sheared out CFRP section remaining bonded to the aluminium adherend as illustrated in Figure 4b.

The relationship between these failure modes and joint strength has been discussed in the literature and is shown in Figure 4c [21]. As the strength of the bond increases, the joint strength will increase in a linear relationship while continuing to fail adhesively. This will continue until the failure load induces a sufficient amount of the stress in the CFRP adherend to cause matrix
cracking and delamination. After this the joint strength will decrease slightly until it plateaus at a constant strength.

The C3 joint’s design leads to the aluminium feature having a small shear area, and hence a large shear stress, with a small discontinuity present in the CFRP adherend. As the CFRP adherend is stronger than the aluminium feature, the aluminium feature is sheared off and the C3 joint fails due to adhesive failure as illustrated in Figures 4a and c. The C1 joint’s design improves on the C3 design by increasing the length of the feature in the loading direction and hence its shear area, reducing the shear stress experienced by the feature. As it is narrower in width than the C3 design, the discontinuity present in the CFRP adherend is still small, the feature can resist delamination failure. Post failure Micro CT scans of the bonded region, shown in Figure 5, prove that the loads experienced by the C1 joint are sufficient to cause non critical crack growth in the uppermost 0° ply (point A) as well as some delamination directly adjacent to the feature (point B). The presence of CFRP damage and adhesive failure suggest that the C1 joint represents the optimum joint strength achievable for this layup as shown in Figure 4c.

The C2 design extends the width of the feature transverse to the loading direction to almost the entirety of the bond width, while maintaining a small length. It was believed that this
extended width would inhibit relative adherend displacement allowing the C2 joint’s feature to act as a crack arrestor, preventing cracks propagating freely through the bondline and creating secondary loading bearing mechanisms. This is discussed more thoroughly in Section 3.2. Due to the feature having a larger shear area and a reduction in fibre volume surrounding the interlocking feature, as compared to C1 or C3, the maximum shear strength of the C2 joint is not sufficient to cause shearing of the aluminium. Instead, the load transfer causes crack growth through the central 90° plies. Upon reaching the ply boundary, this crack causes delamination which runs to the free end of the CFRP adherend causing failure of the joint and leads to the section of CFRP adherend behind the feature being pulled out. The C4 plane strain joint was used to allow direct observation of this failure process. The matrix cracks and subsequent delaminations can be seen in Figure 6 at points A and B, respectively, with the pulled out section highlighted at the left of the picture. The adhesive layer between the sheared out CFRP adherend section and the aluminium adherend does not undergo complete failure with the removed CFRP section remaining bonded to the aluminium adherend as can be seen at point C in Figure 6.

![Figure 6. SEM image of a configuration 4 (C4) plane strain joint showing delamination failure](image)

**3.2 Work to Failure**

Figure 7 shows that there is an increase in work to failure for all interlocking joints over a reference flat joint. The improvements in work to failure follow the same trend as the strength improvements with the greatest improvement coming from C1 followed by C2 and C3. C1 and C3 joints fail catastrophically with no secondary load bearing mechanisms present before total failure of the adhesive bondline. As the profile designs in joints C1 and C3 have a small feature width as compared to the bond width, the feature cannot arrest crack growth through the adhesive layer. Therefore, cracks which initiate at the free end of the aluminium adherend and propagate freely through the bondline with little time passing between crack initiation and failure. In this way their initial failure is identical to the reference joints (C0) albeit at a higher load. However, unlike flat joints, there are additional load bearing mechanisms present in joints C1 and C3 following adhesive failure. These mechanisms, which occur after the total failure of the adhesive bondline, are illustrated by the dashed lines in Figure 8a. In these cases, although the bond line has failed and is no longer holding the adherends together, the grips constrain adherend motion normal to the bond surface preventing the adherends from separating. In reality, a structural joint utilising this joining technology would undergo disassembly at this point. However, due to the profiles remaining interlocked, the feature must be entirely sheared.
Figure 7. Average work to failure for joint configurations 0-3

Figure 8. a) Load-displacement curves for joint configurations 0-3, b) Load-displacement curve for joint configuration 3. Solid lines represent joint behaviour before total adhesive failure with dashed lines representing behaviour post total adhesive failure.

off before the joint loses all its ability to withstand loading. This shearing vastly increases the joint displacement necessary to completely deconstruct the joint and would cause a large increase in work to failure. Therefore, for the purpose of this work total joint failure is considered to have occurred at the point where the entirety of the bondline has been destroyed and is no longer capable of carrying significant load. Therefore it can be said that the increase in work to failure comes almost entirely from the increase in joint strength, as is shown by the solid lines in Figure 8a.

The C2 joint feature is far wider, occupying the majority of the bond width. Due to the increased width it was believed that the C2 feature would act as a crack arrestsor and create
secondary load bearing mechanisms. Unfortunately, while a secondary mechanism does exist with a small secondary loading step present at \(\approx 270\)N, shown in Figure 8b, it is far less significant than expected. As was mentioned previously, failure of the C2 joint is not caused by total adhesive failure, as is seen in joints C1 and C3. Instead failure is due to shear out of the CFRP region between the free end of the CFRP adherend and the interlocking feature (this shear out region can be seen in the close up in Figure 9).

![Figure 9. SEM image showing differing failure types in the plies surrounding the interlocking feature in joint configuration 2 (C2)](image)

However, the fibres surrounding the feature remain intact after the joints initial failure. It is these two regions surrounding the feature which are responsible for the brief secondary loading region. Unfortunately, these fibres cannot withstand the applied load and quickly fail. This failure is caused by crack growth through the plies in the fibre direction. In the 0° plies, these cracks originate at the edge of the feature and propagate to the free end of the adherend allowing the region directly behind the feature to shear out as shown in close up in Figure 9. In the 90° plies, these cracks propagate transverse to the loading direction to the edge of the adherend as highlighted by the arrows in Figure 9. The entirety of the 90° ply closest to the bondline in all cases remains attached to the sheared out CFRP section. The middle 90° plies, which experience transverse cracking in all cases, can either remain attached to the CFRP adherend or to the sheared out section as seen in Figure 9.

Although, the CFRP layup used in this work was not sufficiently strong enough to withstand the secondary load bearing mechanism, the presence of the additional step is promising and will be pursued in future work. With a more robust CFRP layup, utilising a woven fibre architecture, the secondary load bearing mechanism may be extended sufficiently to have a non trivial effect, creating an even greater increase in work to failure. These woven CFRP adherends will also be manufactured using a mould in method rather than the laser process used in this work further strengthening the CFRP adherend by reducing fibre damage surrounding the interlocked feature.

4. CONCLUSIONS

The effects of a novel hybrid interlocking joint technology on shear strength, joint failure and work to failure was investigated experimentally. Three designs were investigated with a
focus on the effects of the length and width of the feature on joint performance. Comparisons between the three hybrid joints and standard bonded reference joints have shown positive results. Compared to the reference joints, there are increases in ultimate shear strength of 18%, 10%, and 8% for the C1, C2, and C3 joint. These increases are due to the interlocking features preventing the adherends displacing a sufficient distance to allow the adhesive to reach its failure strain. In order for the joint to fail, some damage must be created in the adherends to allow the sufficient displacement to cause adhesive failure. The manner and severity of damage dictates the scale of improvement experienced by each of the designs.

In joints C1 and C3 this damage manifests as shear off of the interlocking aluminium feature. As the feature begins to undergo shear deformation, the adhesive reaches its failure strain with the entire bondline failing. The C3 joint, with small width and length, fails entirely due to adhesive failure with the feature undergoing shear off and no damage present in the CFRP adherend. However, the C1 joint’s feature is more resistant to shear failure due to its elongated length. This allows it to reach a high enough load to cause non critical damage in the CFRP adherend before adhesive failure occurs. Due to the presence of this damage, it is believed that joint C1 represents the optimum strength for the hybrid joints with this layup. Joint C2, with its small length and elongated width, experiences no shear failure. Instead failure is due to delamination and shear out in the CFRP adherend leading to a reduction in shear strength as compared to joint C1 although still an improvement over joints C0 and C3.

There is also a corresponding increase in the work to failure for each joint design. For joints C1 and C3 this increase is driven solely by the increase in shear strength with no additional energy absorption mechanisms present before total adhesive failure. The C2 joint has a small secondary load bearing mechanism which occurs after the initial joint failure. While this new mechanism is encouraging, it does not occur at a high enough load and is not sustained for a sufficiently long period to cause a significant increase in the work to failure. Therefore the increases in work to failure follow the improvements in shear strength with the C1 joint showing the greatest improvement followed by the C2 and C3 joints.

In future work a woven carbon fibre architecture will be utilised with a mould in manufacturing method. These changes are designed to deliver a CFRP adherend with a greater volume of fibres surrounding the interlocking feature while also reducing any fibre damage which may be caused during the laser manufacturing procedure. This has a high potential to deliver a large increase in work to failure and damage tolerance.

5. ACKNOWLEDGEMENTS

References


MODELLING OF NOVEL HYBRID COMPOSITE-METAL JOINTS

Jordan Jones*, Luiz F.Kawashita †, Byung Chul Kim †, Stephen R.Hallett†

*Bristol Composites Institute, (ACCIS) Advanced Composites Collaboration for Innovation & Science University of Bristol, Queen’s Building, University Walk, Bristol, BS8 1TT, United Kingdom
Jordan.Jones@bristol.ac.uk

†Bristol Composites Institute, (ACCIS) Advanced Composites Collaboration for Innovation & Science University of Bristol, Queen’s Building, University Walk, Bristol, BS8 1TT, United Kingdom
Luiz.Kawashita@bristol.ac.uk, B.C.Eric.Kim@bristol.ac.uk, Stephen.Hallett@bristol.ac.uk

Keywords: Hybrid Structures, Filament Winding, Joint Design.

Summary: This paper explores the development of an integral composite-metal joining technique through the use of novel finite element analysis, for application to an axially loaded strut. The strut contains a metallic end-fitting with pin-like protrusions. Individual carbon fibre tows are then modelled as being wound onto the end-fitting to give a prediction of the as-manufactured geometry for a range of pin and tow configurations. These different configurations can then be further modelled under mechanical loading for prediction of their relative performance ranking.

1. Introduction

A key requirement for aircraft structures is the minimisation of component weight. This can be achieved through the inclusion of composite materials in the design of hybrid structures. The considered application is a tubular metallic structure which predominately experiences axial loads (strut) (see Figure 1a).

The aim of this research is to assess the feasibility of substituting the central part of the strut with carbon-fibre composite. The strut would then comprise of two metallic end-fittings with pin-lug joints connected via a mechanical form fit provided by the winding of carbon-fibre filaments around protrusions (pins) structured onto the surface of the metal (see Figure 1b). The pin-lug joints transmit bearing loads to the metallic end-fitting but the composite component encounters mainly planar loading (tension and compression).

Note that a cylindrical connector tube is initially situated between the two end-fittings in order provide a winding surface for the fibre-tows. This can either be removed post fabrication or maintained to improve strut properties in compression (beyond the scope of this work).

Joining methods which consider sculpting pins onto the surface of the metal in the joint region have been examined in previous work. Ucsnik et al. [1] conducted an experimental investigation into CFRP-steel double lap shear joints. A range of pin shapes were manufactured onto a steel plate, over which biaxial carbon non-crimp fabric (NCF) was then stacked prior to vacuum assisted resin infiltration. Improvements in joint strength and failure behaviour were...
observed against a reference case in which the connection was provided solely through adhesive bonding resulting from resin usage.

The T-IGEL® connection technology developed by TEUFELBERGER Ges.m.b.H [2] uses composite fabrication techniques such as braiding and filament winding to integrate a fibre framework to a pin-structured metallic body, prior to consolidation via resin-transfer-moulding (RTM). The ability to automatically vary the pin geometry and size allows for the possibility of tailoring these parameters to meet loading specifications.

In the present work, filament winding was determined to be the most appropriate method for integrating the composite and metal as it allows for precise control over the fibre paths in the component. Each fibre-tow is placed individually with a repetition of the winding pattern until complete coverage of the mandrel has been achieved. In comparison, braiding achieves full coverage with a single pass of the mandrel and involves complex interactions between the braid-yarns and braiding ring, whose frictional forces may cause significant deviation from the intended path (braiding angle). Greater precision during layup of the fibre tow is sought in order to allow for the positioning of the tow around the pins. This is important with regards to avoiding puncturing of the fibre-tow and the associated fibre breakage and strength degradation.

The work presented here focuses on a modelling approach that is implemented in LS-DYNA [3] to simulate the winding of carbon fibre tows around the metallic end-fitting and pin configurations in order to establish the composite-metal connection. In accordance with the Digital Element (DE) Method [4], the tow is modelled as a chain of truss elements with circular cross-section. Each element in the chain is described as a "digital element". As the digital element
length approaches zero, the chain becomes fully flexible, allowing it to imitate the flexural behaviour of a real tow or yarn.

Following this, joint tension is simulated. A discussion of tow stress and pin shear force is then provided for two end-fitting and pin configurations in order to ascertain their relative mechanical performance.

2. Modelling Approach

The full strut (consisting of both metallic end-fittings joined by a cylindrical connector tube, see Figure 1), behaves analogously to the mandrel in the winding process. That is, it is this full structure which rotates about its longitudinal axis (as will described in the next section). However, for convenience, throughout this discourse the cylindrical connector tube is referred to as the mandrel.

The metallic end-fittings, pins and mandrel were constructed using shell elements and were assigned rigid material properties. This was sufficient as only a contact surface was required at this stage to study the effect of the inter-tow and tow-metal interactions on determining the path of the tow on the structure. The pin-lug connectors were omitted from the model, as they have no effect with regards to the joining mechanism proposed.

A cylindrical end-fitting was constructed with dimensions as listed in Table 1. The pins were also assumed to be cylindrical in shape, so only definition of their diameter and length was required. The diameter of the mandrel is equal to the end-fitting diameter by default. Note that a relatively short mandrel length was used in order to minimise computational expense. A longer mandrel necessitates a longer virtual tow and therefore a greater number of truss elements are required in the simulation.

Table 1. Strut dimensions for winding simulation (all units mm)

<table>
<thead>
<tr>
<th></th>
<th>End-fitting(s)</th>
<th>Pin(s)</th>
<th>Mandrel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer Diameter</td>
<td>60</td>
<td>3</td>
<td>52.5</td>
</tr>
<tr>
<td>Height</td>
<td>35</td>
<td></td>
<td>5</td>
</tr>
</tbody>
</table>

Two pin distributions were constructed. The first was produced according to the maximum pin density. In this case, pin separation was at its minimum allowable and was determined by some factor of the tow diameter. This configuration has 31 pins positioned around the circumference of the end-fitting, (i.e. 31 pins per row), and 5 rows of pins (see Figure 2a).

Case 2 was established through use of an analytical process which aimed to improve loading of the pin array by more evenly distributing this load across the each of the pins. More equal loading of the pins is preferred as it may reduce the possibility of premature joint failure. The resulting configuration has 26 pins per row and 5 rows of pins (Figure 2b).

A virtual tow of an initial length equal to three digital elements was built. An end-node was assigned as the source node of the *ELEMENT_BEAM_SOURCE (EBS) function. EBS allows for the automatic generation of truss elements at the specified source node. The opposing
end-node was constrained to the surface of the strut. As it rotated during the winding simulation, the end-node of the virtual tow was constrained to move in sequence. The action of this rotation drew out the truss elements of the virtual tow from the specified source node. Thus giving the appearance of a tow being passed through the delivery eye of a filament winder and onto the structure.

No attempt was made to replicate the physical properties of either the tow or the metal. Material properties were assigned to ensure sufficient contact definition and reasonable analysis times.

3. Winding Pattern Determination

The winding of the tow around the pin arrays constructed on the metallic end-fittings was determined so as to allow the tow to be positioned around the pins. Determination of the tow path in these regions is dependant on the diameters of the pins and there relative spacing. Variation in the path can be achieved by specifying the number of pins around which the tow must turn prior to reversing direction.

For characterisation of the tow path down the mandrel length, (or more precisely between the pin arrays on the end-fittings at each end of the strut), it is necessary to specify a desired winding angle, $\theta$. This is the acute angle given by the tow trajectory and the mandrel’s longitudinal axis. The ability to pre-determine the winding angle may be limited by the the presence of the pins and the pre-condition established to avoid tow puncturing.

Consequently, a nearest-pin-matching algorithm was employed to find the closest possible winding angle that can be achieved whilst maintaining a tow trajectory that passes through the pin array. This angle is denoted by $\theta_d$, where the subscript $d$ denotes the path of the tow "down" the mandrel.

It is necessary to ensure that full pin coverage is produced by the winding process. That is, the tow travels through all the channels created by the construction of the pin array and is therefore in contact with each of the pins. For this to hold, irrespective of the pin array and the manner of winding; the winding pattern must be constructed so that each pin is wound in turn, with the tow moving sequentially around the end-fitting onto the next pin (in the row).
results in the definition of effectively two winding angles, $\theta_d$ and $\theta_u$ (the subscript $u$ denotes the "up" direction), where the latter is determined by the need to wind the tow onto the next pin in the array.

A winding angle of $10^\circ$ was specified for the two end-fitting configurations considered. Table 2 displays the resulting values of $\theta_u$ and $\theta_d$ calculated for the two cases. The tow was further designated to turn around a single pin at the top of the structure.

Table 2. Strut dimensions for winding simulation

<table>
<thead>
<tr>
<th></th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Pins Per Row</td>
<td>31</td>
<td>26</td>
</tr>
<tr>
<td>$\theta_d$</td>
<td>11.4$^\circ$</td>
<td>11.4$^\circ$</td>
</tr>
<tr>
<td>$\theta_u$</td>
<td>5.7$^\circ$</td>
<td>0$^\circ$</td>
</tr>
</tbody>
</table>

Once a single repeatable "unit" of the desired winding pattern has been identified, it can then iterated for the number of pins in the circumferential direction, to provide replication of a full wind layer. Note that complete coverage is not guaranteed as with conventional winding in which the tow path is offset by the tow (band) width at each repetition. In this application, the degree of offset is determined by the pin separation which may be significantly larger.

Motion of the source node (which represents the tow delivery eye) is prescribed using a displacement vs time curve. The rigid bodies which compose the mandrel, end-fittings and pins are merged so that the prescribed rotational velocities act uniformly. Figure 3 displays the path of the tow on the structure for cases 1 and 2.

![Figure 3. Single filament-wound layer with tow (red), end-fittings and pins (grey), Mandrel (blue)](image)
4. Virtual Testing

To be able to conduct virtual tests, it is first necessary to alter the global structure of the virtual tow. This is accomplished by extracting the nodal coordinates that represent the geometric centres of the (truss) digital elements in the virtual tow, at the end of the winding simulation. These define a centreline path of the tow on the structure. The virtual tow is then made to form a closed-loop through the merging of the first and final nodes in the digital-element-chain. The aim of the reconfiguration of the virtual tow is to provide a simple mechanism to allow for maintenance of tension in the tow under loading.

The truss elements in the virtual tow were assigned a modulus of 238 GPa, (equivalent to a high modulus 12k carbon fibre tow [5]). The pins were replaced with beam elements of equal diameter and length. This allowed for extraction of the resultant shear force exerted on the pins without the additional computational expense incurred through inclusion of realistic material properties for the metallic elements.

Tension was simulated by prescribing a displacement to the end-fittings at either end of the strut. The displacement direction is along the strut’s longitudinal axis, with a total magnitude of displacement of 2 mm, with each end-fitting moving 1 mm in opposing directions (positive and negative). The total time in the explicit finite element analysis was 10 seconds, with a linear displacement increase up to the maximum displacement at 5 seconds, at which point the end-fittings’ position was maintained for a further 5 seconds. This was to reduce any oscillatory behaviour in the virtual tow.

Resin properties were excluded from the model as the fibre-tows form the primary load carrying constituent of the composite material, and therefore contribution by the resin to ultimate tensile strength of the joint is expected to be negligible. The virtual tests were thereby considered analogous to the loading of a strut connected via a dry fibre-tow network (rather than the final cured component).

4.1 Tow Analysis

The axial force (force along the truss element’s longitudinal axis) was extracted from the model. This was averaged at each 0.1 second increment for the total number of truss elements which make up the virtual tow. An estimate of the stress in the tow resulting from joint tension was gained through a division by the cross-sectional area of the truss element(s).

Figure 4 shows the tow stress vs time curve for both of the cases considered. Horizontal lines are drawn at values which represent the average stress between times 5 and 10 seconds. These values are given by 2518 MPa and 3388 MPa for cases 1 and 2 respectively. The tow in case 2 experiences a 35% greater average stress than case 1. This is due to the fact that this case has fewer pins on the structure, resulting in reduced tow density.

Figure 5 displays the variation in the (axial) force in the tow structure for case 2 at time 10 seconds. Clearly, the tow is subjected to the largest tensile forces in regions where it traverses down the mandrel as no force is taken up by friction with pins at this point.
4.2 Pin Analysis

Analysis of a single end-fitting for each configuration was conducted. The pin rows were numbered sequentially from 1 to 5, with the first pin row denoting the row nearest to the mandrel.

Figure 6 displays the average resultant shear force (N) per pin for each row against analysis time. Also plotted are horizontal lines representing the force values averaged between the times 5 and 10 seconds; where the displacement of the end-fittings were held at the maximum. Their corresponding values are displayed in Table 3.
In both cases the loading of pins in the middle rows 2-4 is minimal in comparison to the force exerted on the pins in rows 1 and 5, indicating that for the single layer wind conducted here, they do not contribute to load carrying capacity of the pin array.

Case 2 would seem to provide an improved load distribution across pin rows 1 and 5 of the array. The resultant shear force in row 5 is only 18% larger than for row 1. Where as for case 1, this difference is approximately 45%.
Table 3. Average resultant shear force between times 5 and 10 seconds

<table>
<thead>
<tr>
<th>Pin Row</th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>257.3</td>
<td>247.9</td>
</tr>
<tr>
<td>2</td>
<td>20.1</td>
<td>20.2</td>
</tr>
<tr>
<td>3</td>
<td>19.0</td>
<td>19.8</td>
</tr>
<tr>
<td>4</td>
<td>16.5</td>
<td>20.5</td>
</tr>
<tr>
<td>5</td>
<td>372.8</td>
<td>291.8</td>
</tr>
</tbody>
</table>

Figure 7. Two filament-wound layers defined using different winding angles $\theta$. The complete strut would have a balanced layup to avoid coupling effects.

5. Conclusion

A methodology for the construction of a novel hybrid composite-metal joint has been demonstrated. A filament winding simulation was conducted to predict the as-manufactured tow position on the structure under the precondition of precise tow placement around the pin array.

A particular winding pattern was constructed on two example end-fitting and pin configurations, which were then examined with regards to the shear force exerted on the pins due to joint tension. It was found that the relative spacing of the pins in the array has a significant effect on the magnitude and distribution of the resulting shear force in the pins, and on the estimated stress in the tow. This analysis provides scope for the consideration of optimal pin distributions and winding patterns in order to maximise joint properties according to its loading criterion. Further work will consider this objective for the case of a strut with a more realistic degree of tow density, i.e. multiple filament wound layers (see Figure 7).

Acknowledgements

This work was supported by Airbus UK and the EPSRC through an iCASE studentship.
References


[2] Teufelberger Composite Ges.m.b.H. Vogelweiderstr. 50, 4600 Wels, Austria


A MIXED-MODE COHESIVE MODEL FOR DELAMINATION WITH ISOTROPIC DAMAGE AND INTERNAL FRICTION

Federica Confalonieri*, Umberto Perego*

*Politecnico di Milano
Piazza Leonardo da Vinci, 32
federica.confalonieri@polimi.it, umberto.perego@polimi.it

Keywords: Mixed-mode delamination, cohesive model

Summary: This work deals with the formulation of a thermodynamically consistent, isotropic damage cohesive model for mixed-mode delamination under variable mode ratio. The proposed model is based on the introduction of an internal friction angle in the tensile case, that allows for an accurate modelling of the interaction between normal and shear openings.

1. INTRODUCTION

Delamination is a common failure mechanism in laminated composite materials, often characterized by mixed-mode loading conditions with variable mode ratio. The capability to accurately predict the progressive mixed-mode delamination in real life engineering applications is a key ingredient in the development of robust design tools. The computational strategies for the numerical simulation of the delamination process are often based on the use of interface elements within the cohesive zone approach (see, for instance, [1, 2, 3]). Among the main difficulties arising in the definition of a robust cohesive model, there are, on the one hand, the necessity to ensure its accuracy and thermodynamic consistency for arbitrary loading paths and, on the other hand, the capability to correctly reproduce the typical growth of fracture energy in passing from Pure Mode I to Pure Mode II. A number of experimental works on composite materials (see e.g. [4] for a comprehensive literature survey) show that the fracture energy in Pure Mode II is typically higher than in Pure Mode I, as a result of a change in the involved micromechanical mechanisms, with a transition from pure mode I loading characterized by matrix cleavage and fiber pull-out, to mode II conditions dominated by the formations of cusps and hackles [5, 6].

A new cohesive model, specifically conceived for mixed mode delamination with variable mode ratio, is presented in this work. The model is based on an isotropic damage formulation and is thermodynamically consistent. The coupling between the normal and the shear behaviour is achieved by projecting the cohesive stress vector onto three unit vectors defining three distinct damage modes, accounting for mixed-mode interaction through a parameter playing the role of an angle of internal friction. The proposed model is able to capture the non-monotonic increase of fracture energy at increasing mode ratios without the need of introducing any empirical law, as demonstrated by the comparison between the outcome of the present cohesive law and the data resulting from experimental tests performed with the Mixed Mode Bending test [7] apparatus.
2. FORMULATION

The coupling between the normal and the shear behaviour is achieved by defining the three damage modes, represented in Figure 1, in the plane of non-dimensional cohesive tractions (i.e. the cohesive stresses divided by the corresponding pure Mode peak values). Each damage mode is identified by a unit normal vector \( n_i \) depending on an angle \( \alpha \) playing the role of an angle of internal friction. The first damage mode is associated to an opening dominated decohesion process, while the second and third damage modes are the result of the interaction between shear and normal relative displacements. The unit vector \( n^1 \) is associated to the opening-dominated mode, while \( n^2 \) and \( n^3 \) define the two shear-dominated, symmetrical modes.

\[
\begin{align*}
n^1 &= [1 \ 0] \\
n^2 &= [\sin \alpha \ \cos \alpha] \\
n^3 &= [\sin \alpha - \cos \alpha]
\end{align*}
\] (1)

The modelling of the delamination under mixed-mode conditions with variable mode ratios and arbitrary loading paths is obtained formulating an isotropic damage cohesive model in a thermodynamically consistent framework. The free energy per unit surface \( \Psi \) is defined as:

\[
\Psi = \frac{1}{2} K (\langle \delta^n \rangle_-)^2 + \frac{1}{2} (1 - d) K (\langle \delta^n \rangle_+)^2 + \frac{1}{2} (1 - d) K (\delta^s)^2
\] (2)

being \( \delta^n \) and \( \delta^s \) the normal and tangential relative displacements, \( K \) the elastic stiffness of the interface and \( d \) the isotropic damage variable. The unilateral effect is accounted for by the introduction of the Macauley brackets \( \langle \rangle \) that allow to distinguish between the tensile and the compressive cases. Within a classical thermodynamic framework, the cohesive tractions \( t^n \) and \( t^s \) and the strain energy per unit of damage growth \( Y \) can be defined by means of the following state equations:
\[ t^n = \frac{\partial \Psi}{\partial \delta^n} = K\langle \delta^n \rangle_- + (1 - d) K\langle \delta^n \rangle_+ \quad (3) \]
\[ t^s = \frac{\partial \Psi}{\partial \delta^s} = (1 - d) K\delta^s = (1 - d) \frac{t^n_0}{\delta^n_0} \delta^s \quad (4) \]
\[ Y = - \frac{\partial \Psi}{\partial d} = \frac{1}{2} K \left( \langle \delta^n \rangle_+ \right)^2 + \frac{1}{2} K \left( \delta^s \right)^2 \quad (5) \]

with \( t^n_0 = K\delta^n_0 \) and \( t^s_0 = K\delta^s_0 \), being \( t^n_0 \) and \( t^s_0 \) the strengths in pure Modes I and II and \( \delta^n_0 \) and \( \delta^s_0 \) the relative displacements at the onset of delamination, i.e. corresponding to \( t^n_0 \) and \( t^s_0 \). For the sake of simplicity, only the tensile case, i.e. \( \delta^n \geq 0 \), will be considered in the following.

Let us introduce the vector \( \mathbf{t} \), collecting the non-dimensional cohesive tractions, i.e:
\[ \mathbf{t} = \begin{bmatrix} t^n \\ t^s \end{bmatrix}^T = \begin{bmatrix} t^n_0 \\ t^n_0 \\ t^s_0 \end{bmatrix}^T \quad (6) \]

A vector of effective cohesive stresses \( \mathbf{s} \) is introduced to account for the interaction between normal and tangential behaviour: each component \( s^i \), \( i = 1, 2, 3 \) is computed by projecting \( \mathbf{t} \) onto the direction defined by one of the three normals in eqn. 1, identifying a distinct damage mode:
\[ s^1 = \mathbf{t}^T \mathbf{n}^1 = \mathbf{t}^n \]
\[ s^2 = \mathbf{t}^T \mathbf{n}^2 = \mathbf{t}^n \sin \alpha + \mathbf{t}^s \cos \alpha \quad (7) \]
\[ s^3 = \mathbf{t}^T \mathbf{n}^3 = \mathbf{t}^n \sin \alpha - \mathbf{t}^s \cos \alpha \]

In matrix form:
\[ \mathbf{s} = \mathbf{N} \mathbf{t} \quad (8) \]

where matrix \( \mathbf{N} = [\mathbf{n}^1 \mathbf{n}^2 \mathbf{n}^3]^T \) gathers the unit vectors \( \mathbf{n}^i \). The same approach can be followed also for the relative displacements. Let \( \mathbf{\delta} \) define the vector collecting the non-dimensional opening displacements in the normal and in the tangential directions as:
\[ \mathbf{\delta} = \begin{bmatrix} \delta^n \\ \delta^s \end{bmatrix}^T = \begin{bmatrix} \delta^n_0 \\ \delta^n_0 \\ \delta^s_0 \end{bmatrix}^T \quad (9) \]

A vector \( \mathbf{w} \) of effective relative displacements, conjugated to the effective cohesive stresses \( \mathbf{s} \) in the expression of the free energy density \( \Psi \), is defined by introducing three structural vectors \( \mathbf{m}^i \), defined as:
\[ \mathbf{m}^1 = a \begin{bmatrix} 1 \\ 0 \end{bmatrix}^T \]
\[ \mathbf{m}^2 = b \begin{bmatrix} \sin \theta & \cos \theta \end{bmatrix}^T \]
\[ \mathbf{m}^3 = b \begin{bmatrix} \sin \theta & -\cos \theta \end{bmatrix}^T \quad (10) \]
As depicted in figure 1, \( m^1 \) is aligned to \( n^1 \) for symmetry considerations, while the orientation of \( m^2 \) and \( m^3 \) is defined as a function of the angle \( \theta \), different from the angle of internal friction \( \alpha \). The effective relative displacements \( w^i, i = 1, 2, 3 \) are obtained by the projection of the non-dimensional relative displacements vector \( \delta \) onto a structural vector \( m^i \) as:

\[
\begin{align*}
 w^1 &= \delta^T m^1 \\
 w^2 &= \delta^T m^2 \\
 w^3 &= \delta^T m^3
\end{align*}
\]

In matrix form:

\[
 w = M \delta
\]

being \( M \) the matrix gathering the components of the three structural vectors \( m^i \).

The strain energy \( \Psi \) can be expressed as a function of the effective quantities defined in eqns. 7 and 11 to underline the contribution of each single damage mode.

\[
 \Psi = \frac{1}{2} t^n \delta^n + \frac{1}{2} s^2 w^2 + \frac{1}{2} s^3 w^3
\]

The two unknown constants \( a \) and \( b \) are, then, determined by imposing that \( \Psi \) remains the same in passing from the direct to the effective variables, i.e.

\[
\frac{1}{2} (t^n \delta^n + t^s \delta^s) = \frac{1}{2} \left( s^1 w^1 + s^2 w^2 + s^3 w^3 \right)
\]

thus, obtaining:

\[
\begin{align*}
 a &= (t^0_0 \delta^n_0 - t^0_0 \delta^s_0 \tan \alpha \tan \theta) \\
 b &= \frac{t^0_0 \delta^n_0}{2 \cos \alpha \cos \theta}
\end{align*}
\]

By considering the strain energy \( \Psi^i \) associated to each damage mode, the following state equations define three effective strain energies release rates \( Y^i \):

\[
\begin{align*}
 Y^1 &= -\frac{\partial \Psi^1}{\partial d} = \frac{1}{2} \left( t^n_0 \delta^n_0 - t^s_0 \delta^s_0 \tan \alpha \tan \theta \right) \left( \delta^n \right)^2 \\
 Y^2 &= -\frac{\partial \Psi^2}{\partial d} = \frac{1}{4} t^s_0 \delta^n_0 \left[ \tan \alpha \tan \theta \left( \delta^n \right)^2 + \left( \delta^s \right)^2 + (\tan \alpha + \tan \theta) \delta^n \delta^s \right] \\
 Y^3 &= -\frac{\partial \Psi^3}{\partial d} = \frac{1}{4} t^s_0 \delta^n_0 \left[ \tan \alpha \tan \theta \left( \delta^n \right)^2 + \left( \delta^s \right)^2 - (\tan \alpha + \tan \theta) \delta^n \delta^s \right]
\end{align*}
\]

that play the role of driving forces acting on the corresponding damage modes, with:

\[
 Y^1 + Y^2 + Y^3 = \frac{1}{2} t^n_0 \delta^n_0 \left( \delta^n \right)^2 + \frac{1}{2} t^s_0 \delta^s \left( \delta^s \right)^2 = Y \geq 0
\]
In the following, it is assumed that:

\[ \theta = \arctan \left( \frac{t_n 0 \delta_n 0}{t_s 0 \delta_s 0} \tan \alpha \right) \] (18)

so that in the tensile case \( Y^1 \geq 0 \) for \( \alpha \leq 45^\circ \). On the contrary, either \( Y^2 \) or \( Y^3 \) can be negative, but their sum \( Y^2 + Y^3 \), representing the fraction of the strain energy release rate associated to the shear-dominated damage modes, is always non-negative.

The damage activation criterion is written as a function of the strain energy release rates \( Y^i \) as:

\[
\phi = \left( \frac{Y^1}{\chi^0_0 + \chi^1} \right)^k + H \left( Y^2 \right) \left( \frac{Y^2}{\chi^2_0 + \chi^2} \right)^k + H \left( Y^3 \right) \left( \frac{Y^3}{\chi^3_0 + \chi^3} \right)^k - 1 \leq 0
\] (19)

being the exponent \( k \) an input parameter of the model, \( H() \) the Heaviside function, introduced to avoid negative contributions of \( Y^2 \) and \( Y^3 \) to the damage activation function, and \( \chi^i + \chi^0_i \) the current threshold for the \( i \)-th damage mode, evolving during the decohesion process as a function of the damage variable \( d \). \( \chi^0_i \) represents the initial threshold, while \( \chi^i \) is the internal variable, governing the threshold evolution for increasing damage and determines the shape of the softening branch. Figure 2 shows the damage activation surface at the onset of decohesion, for increasing values of the internal friction angle \( \alpha \) and \( k = 4 \) (a) and for increasing values of the exponent \( k \), while maintaining a constant value of the angle \( \alpha = 30^\circ \) (b).

In the framework of a classical thermodynamic formulation, the cohesive model is completed by the introduction of the damage evolution law and of the Kuhn-Tucker conditions. The damage rate is obtained by imposing that \( \varphi = 0 \) and \( \dot{\varphi} = 0 \), i.e.:

\[
\dot{d} = - \frac{\partial \phi}{\partial d} \dot{\delta}^n + \frac{\partial \phi}{\partial d} \dot{\delta}^s = \sum_{i=1}^{3} \left( \frac{\partial \phi}{\partial Y^i} \frac{\partial Y^i}{\partial \delta} \right) \dot{\delta}^n + \sum_{i=1}^{3} \left( \frac{\partial \phi}{\partial \chi^i} \frac{\partial \chi^i}{\partial d} \right) \dot{\delta}^s
\] (20)
while the loading-unloading conditions read as:

\[ \phi \leq 0 \quad \dot{\phi} \geq 0 \quad \phi \dot{\phi} = 0 \] (21)

The thermodynamic consistency of the cohesive model can be proven considering the Clausius-Duhem inequality for isothermal processes and showing that the mechanical dissipation is always non negative.

\[ D = Y^1 \dot{\delta} + Y^2 \dot{\delta} + Y^3 \dot{\delta} = (Y^1 + Y^2 + Y^3) \dot{\delta} = \dot{Y} \geq 0 \] (22)

The expressions of \( \chi^0 \) and \( \chi^i \) can be found by prescribing a fixed functional form for the two traction-separation laws in pure Modes. In this work, the case of a bilinear shape, shown in Figure 3, is envisaged. Under this hypothesis, the two branches of the pure Mode traction-separation law depicted in Figure 4 can be described as:

\[ t = t_0 \delta \quad \text{for} \quad \delta \leq \delta_0 \] (23)

\[ t = (1 - d) t_0 \delta = t_0 \frac{\delta_{cr} - \delta}{\delta_{cr} - \delta_0} \quad \text{for} \quad \delta \geq \delta_0 \] (24)

From eqn. 24 it is possible to derive the relation between the damage variable \( d \) and the relative displacement \( \delta \) for \( \delta \geq \delta_0 \), i.e.

\[ \delta = \frac{\delta_{cr} \delta_0}{\delta_{cr} - (\delta_{cr} - \delta_0) d} \] (25)

Pure Mode I and II conditions can be retrieved by imposing that:

\[ \delta^n \neq 0 \quad \delta^s = 0 \quad \text{for pure Mode I} \] (26)

\[ \delta^n = 0 \quad \delta^s \neq 0 \quad \text{for pure Mode II} \] (27)
The effective strain energies per unit of damage growth under pure Mode I and II loading conditions can be found by substituting eqns. 27 into eqn. 16:

\[
\begin{align*}
Y_{1mI} & = \frac{1}{2} \left( t_0^n \delta_0^n - t_0^s \delta_0^s \tan \alpha \tan \theta \right) \left( \frac{\delta}{\delta} \right)^2 \\
Y_{2mI} & = \frac{1}{4} t_0^s \delta_0^s \left( \tan \alpha \tan \theta \right) \left( \frac{\delta}{\delta} \right)^s \\
Y_{3mI} & = \frac{1}{4} t_0^s \delta_0^s \left( \tan \alpha \tan \theta \right) \left( \frac{\delta}{\delta} \right)^s \\
Y_{1mII} & = 0 \\
Y_{2mII} & = \frac{1}{4} t_0^s \delta_0^s \left( \frac{\delta}{\delta} \right)^s \\
Y_{3mII} & = \frac{1}{4} t_0^s \delta_0^s \left( \frac{\delta}{\delta} \right)^s 
\end{align*}
\]

with the subscripts \(mI\) and \(mII\) denoting pure Mode I and Mode II loading conditions. It could be noticed that the behaviour in pure Mode II is uncoupled since \(Y_{1mII} = 0\), while in pure Mode I one has \(Y_{2mI} \neq 0\) and \(Y_{3mI} \neq 0\). For that reason, it is necessary to deal first with the pure Mode II case and, then, treat the Pure Mode I case on the basis of the results obtained for Mode II.

Because of the symmetry of the two shear dominated damage modes with respect to the axis \(t_s = 0\), it turns out that \(Y_{2mII} = Y_{3mII}\), \(\chi_2^0 = \chi_3^0\) and \(\chi_2^3 = \chi_3^3\). The initial thresholds \(\chi_2^0\) and \(\chi_3^0\) can be determined by imposing that the activation criterion is fulfilled at the onset of delamination, i.e. \(\varphi = 0\) for \(d = 0\) and \(\delta^s = 1\).

\[
\begin{align*}
\varphi & = \left( \frac{Y_{2mII}^1}{\chi_2^0} \right)^k + \left( \frac{Y_{3mII}^1}{\chi_3^0} \right)^k - 1 = 2 \left( \frac{Y_{2mII}^1}{\chi_2^0} \right)^k - 1 = 0 \quad \Rightarrow \quad \chi_2^0 = \chi_3^0 = 2^{\frac{1}{k}} \frac{1}{4} t_0^s \delta_0^s (30)
\end{align*}
\]

Similarly, the expressions of \(\chi_2^3\) and \(\chi_3^3\) can be found by imposing that the activation criterion is met for a generic point belonging to the softening branch, i.e. that the activation function is zero for \(d > 0\) and \(\delta^s > 1\):

\[
\begin{align*}
\phi & = \left( \frac{Y_{2mII}^3}{\chi_2^0 + \chi_2^k} \right)^k + \left( \frac{Y_{3mII}^3}{\chi_3^0 + \chi_3^k} \right)^k - 1 = 0 \quad \Rightarrow \quad \chi_2^3 = \chi_3^3 = 2^{\frac{1}{k}} \frac{1}{4} t_0^s \delta_0^s \left[ \left( \frac{\delta}{\delta} \right)^2 - 1 \right] (31)
\end{align*}
\]

By substituting eqn. 25 into eqn. 31, one obtains:

\[
\chi_2^3 = \chi_3^3 = 2^{\frac{1}{k}} \frac{1}{4} t_0^s \delta_0^s \left[ \left( \frac{\delta_s}{\delta_s} - \frac{\delta_s}{\delta_s - \delta_0^s} d \right) \right]^2 - \chi_0^2 (32)
\]

Let us now focus on pure Mode I. The expressions of \(\chi_0^1\) and \(\chi_1^3\) can be determined with an analogous procedure, considering the delamination onset and the softening branch, respectively.
For $\delta_n = 1$ and $d = 0$, it holds that:

$$\varphi = \left( \frac{Y_1}{\lambda_0} \right)^2 + \left( \frac{Y_2}{\lambda_0^2} \right)^2 + \left( \frac{Y_3}{\lambda_0^3} \right)^2 - 1 = 0 \quad \Rightarrow \quad \chi_0^1 = \frac{1}{2} \left( \frac{t_n^0 \delta_n^0 - t_s^0 \delta_S^0 \tan \alpha \tan \theta}{1 - (\tan \alpha \tan \theta)^2} \right)^{\frac{1}{2}}$$

(33)

while for a generic non-dimensional opening displacement $\delta_n > 1$ one has:

$$\varphi = \left( \frac{Y_{mI}^1}{\lambda_0^2 + \chi^1} \right)^k + \left( \frac{Y_{mI}^2}{\lambda_0^2 + \chi^2} \right)^k + \left( \frac{Y_{mI}^3}{\lambda_0^3 + \chi^2} \right)^k - 1 = 0$$

(34)

and

$$\chi_1 = \frac{1}{2} \left[ \frac{t_n^0 \delta_n^0 - t_s^0 \delta_S^0 \tan \alpha \tan \theta}{1 - \left( \frac{\delta_n^0 - \delta_S^0}{\delta_n^0 - \delta_S^0} \right)^2 \tan \alpha \tan \theta} \right]^{\frac{1}{2}} - \chi_0^1$$

(35)

The resulting mixed-mode response is depicted in Figure 5. Even though the pure Modes laws are assumed to be bilinear, the softening branch in the mixed-mode law is, in general,
curvilinear, except that for the particular case of identical pure Modes, i.e. \( t^n_0 = t^s_0 = t_0, \)
\( \delta^n_0 = \delta^s_0 = \delta_0 \) and \( \delta^n_{cr} = \delta^s_{cr} = \delta_{cr} \). In this particular case, the resulting mixed-mode traction-separation law can be determined analytically in the case of a radial path, expressing the non-dimensional normal and tangential relative displacements as:
\[
\bar{\delta}^n = (1 - \beta) \bar{\delta} \quad \bar{\delta}^s = \beta \bar{\delta}
\]

By substituting eqns. 36 into eqns. 16 and 19, the activation function turns out to be:
\[
\varphi = \left\{ \frac{(1 - \beta)^2}{\frac{\delta_{cr}}{\delta_{cr} - (\delta_{cr} - \delta_0)d}} \right\}^k + \left\{ \frac{[(1 - \beta) \tan \alpha + \beta]^2 \bar{\delta}^2}{2^{1/k} \left( \frac{\delta_{cr}}{\delta_{cr} - (\delta_{cr} - \delta_0)d} \right)^2} \right\}^k
\]
\[
+ \left\{ \frac{[(1 - \beta) \tan \alpha - \beta]^2 \bar{\delta}^2}{2^{1/k} \left( \frac{\delta_{cr}}{\delta_{cr} - (\delta_{cr} - \delta_0)d} \right)^2} \right\}^k - 1 = 0 \tag{37}
\]

being \( \alpha = \theta \) according to eqn. 18. Solving eqn. 37 for \( d \), the following relationship between the damage variable \( d \) and \( \bar{\delta} \) holds:
\[
d = \frac{\delta_{cr}}{\delta_{cr} - \delta_0} \left( \frac{1}{C_{\beta\alpha} \bar{\delta}} \right) \quad \text{for } \varphi = 0 \quad \text{with}
\]
\[
C_{\beta\alpha} = \left\{ (1 - \beta)^{2k} \left[ 1 - \tan^{2k} \alpha \right] + 0.5 \left[ (1 - \beta) \tan \alpha + \beta \right]^{2k} + 0.5 \left[ (1 - \beta) \tan \alpha - \beta \right]^{2k} \right\}^{\frac{1}{k}} \tag{38}
\]

Thus, from eqns. 3 and 4, for \( \varphi = 0 \) one obtains:
\[
t^n = (1 - \beta) t_0 \left[ -C_{\beta\alpha} \delta_0 \bar{\delta} + \delta_{cr} \right] \tag{39}
\]
\[
t^s = \beta t_0 \left[ -C_{\beta\alpha} \delta_0 \bar{\delta} + \delta_{cr} \right] \tag{40}
\]

### 3. NUMERICAL EXAMPLES

One of the strengths of the proposed cohesive model is that the overall fracture energy is an outcome of the interaction between damage modes without the need of introducing any empirical law and without making any assumption on the loading path. In these numerical examples, the effectiveness of the model in capturing the variation of the fracture energy with the modexity ratio is assessed considering the experimental data of three different fiber reinforced composite materials, namely HMF/5322 I[8], IM7/8552 [9] and AS4/PEEK [10], resulting from Mixed Mode Bending tests [7]. The input parameters required for the definition of the proposed cohesive model, i.e. the fracture energies \( G_{Ic}, G_{IIc} \) and the peak tractions \( t^n_0, t^s_0 \) in pure Modes I and II, the internal friction angle \( \alpha \) and the exponent \( k \) appearing in the activation function \( \phi \),
Federica Confalonieri, Umberto Perego

<table>
<thead>
<tr>
<th>Material</th>
<th>$G_{Ic}$</th>
<th>$G_{IIc}$</th>
<th>$t_n$</th>
<th>$t_s$</th>
<th>$K$</th>
<th>$\alpha$</th>
<th>k</th>
</tr>
</thead>
<tbody>
<tr>
<td>HMF/5322</td>
<td>0.3043</td>
<td>0.8039</td>
<td>10</td>
<td>18</td>
<td>10,000</td>
<td>32</td>
<td>10</td>
</tr>
<tr>
<td>IM7/8552</td>
<td>0.2085</td>
<td>0.7713</td>
<td>60</td>
<td>90</td>
<td>20,000</td>
<td>36</td>
<td>10</td>
</tr>
<tr>
<td>AS4/PEEK</td>
<td>0.969</td>
<td>1.719</td>
<td>75</td>
<td>80</td>
<td>100,000</td>
<td>42.8</td>
<td>1.56</td>
</tr>
</tbody>
</table>

Table 1. Adopted parameters.

Figure 6. HMF/5322. Experimental vs numerical mixed-mode fracture energies. Dots: experimental data [8]. Solid line: result of the present model.

are listed in Table 1. Figures 6, 7 and 8 show the comparison between the experimental data and the curve obtained with the proposed cohesive law: in all the three cases a good agreement can be observed. In addition, for comparison purposes, the empirical Benzeggagh-Kenane (B-K) law [11] $G_c = G_{Ic} + (G_{IIc} - G_{Ic}) \left( \frac{G_{Il}}{G_{Ic} + G_{IIc}} \right)^{\eta}$ is depicted in Figures 7 and 8, considering the exponent $\eta$ determined in [9] and [10] by a fitting of the experimental data.

4. CONCLUSIONS

A new isotropic damage cohesive model for the simulation of mixed-mode delamination has been presented in this work. The proposed model is based on the introduction of a dissipation mechanism described by a parameter qualitatively similar to an angle of internal friction, leading in a natural way to a coupling between normal and shear behaviours. The model is thermodynamically consistent and is able to accurately reproduce the fracture energy under mixed-mode loading conditions, as shown in the numerical examples.

5. Acknowledgements

The financial support of Tetra Pak Packaging Solutions is gratefully acknowledged.

References


MULTI-SCALE MODELING OF THE STRUCTURAL AND VIBRATIONAL BEHAVIOR OF CARBON NANOTUBE REINFORCED POLYMERIC NANOCOMPOSITE PLATES

S. Doagou-Rad1,2,*, J. S. Jensen1, A. Islam1,2

*1Centre of Acoustic-Mechanical Micro Systems (CAMM), Technical University of Denmark
Building 352, DK-2800 Kgs. Lyngby, Denmark
sadora@mek.dtu.dk

2 Department of Mechanical Engineering, Technical University of Denmark
Produktionstorvet, Building 427A, DK-2800 Kgs. Lyngby, Denmark
json@elektro.dtu.dk
mais@mek.dtu.dk

Key words: Multi-scale Modeling, Experimental validation, Nanocomposites.

Summary: Polymeric nanocomposites reinforced with carbon nanotubes are being considered as alternatives in many industrial applications. However, the mechanical behavior of the industrially produced nanocomposites is yet to be fully understood. In this study, Polyamide 6,6-based nanocomposites reinforced with different contents of multi-walled carbon nanotubes (MWCNTs) were manufactured using an injection moulding process. A multi-scale approach was followed to numerically model the mechanical behavior of the nanostructured materials. In order to find the stiffness matrix of the carbon nanotubes, different loading scenarios were conducted on the tubes using molecular dynamics simulations (LAMMPS). The derived properties of the carbon nanotubes from the atomistic simulations were included in a Benveniste Mori-Tanaka based micromechanical model allowing us to acquire the elastic mechanical properties in the produced nanocomposites with different arrangements and contents of the nanotubes. The numerical results were also compared with the experimental properties of the nanocomposites produced via different processing settings leading to distinct microstructures. Eventually the derived properties and stiffness matrices were incorporated in an in-house finite element code for plate vibrations. The results show how the arrangement and the content of the carbon nanotubes in the injection-moulded nanocomposite plates define their structural and vibrational behavior.

1 INTRODUCTION

Progress in science and technology has created new possibilities and opportunities for different industrial applications. Among these novel possibilities, extraordinary properties of carbon nanotubes (CNTs) have attracted soaring interests of not only research institutes, but also industries especially in the recent years. Since their discovery in 1991, significant research has been underway to study their unique properties [1,2]. In fact, due to their exceptional properties, composites reinforced with CNTs are being considered as viable alternatives to many conventional materials in various industrial applications including transportation, automotive, aerospace, sporting goods, energy, and infrastructure sectors [3].
One of the main reasons for the considerable attention to these nano fillers is that due to their geometry and intrinsic characteristics, their incorporation in the metallic or polymeric matrix can introduce new mechanisms in the composites, and result in new combinations of features, not available elsewhere. In spite of these promising features, their stable and tailored manufacturing as the prerequisite to their effective deployment is still challenging.

Due to several challenges regarding the complete experimental characterization of the nanocomposites reinforced with carbon nanotubes, extensive modeling approaches have been pursued to capture their various behaviors. In fact, because of their size, geometry, and properties, experimental characterization of the tubes has been quite challenging and expensive. Moreover, due to presence of a wide range of these nanofillers in respect of geometry, defects, and method of production, etc. the reported experimental results have been quite different. For instance, a range of results from 300 GPa-1.4 TPa have been reported for longitudinal elastic modulus of the carbon nanotubes. Therefore, simulation of the carbon nanotubes has been an area of interest. Especially, when macro scale properties of the nanocomposites are required, simulation methods such as molecular dynamics (MD) or molecular mechanics (MM) can be quite helpful to find the properties in the nanoscale.

Like other polymeric composites, the final properties of composites are defined as a function of arrangement and orientation of fibers. However, due to the high interfacial energy and aspect ratio of the nanotubes, arrangement control of the nanotubes in the polymeric matrix is a challenging task. Moreover, most efforts in this area have been limited to the bounds of laboratories. However, production of the nanocomposite components using injection moulding and microinjection molding provides a powerful tool to tune the properties toward the desired microstructure within the composite. Modulation of the involved parameters in the manufacturing process along with the mold design can lead to tailored polymeric composites. The properties of the nanocomposites are effectively influenced by the distribution, dispersion, alignment and the interfacial properties of MWCNTs in the polymer system. Thus, parameters such as injection speed, injection pressure, melting temperature, and mould temperature vary the properties of the nanocomposites drastically [4–6].

The purpose of this study is to investigate of the influence of the alignment and randomness of the carbon nanotubes on the mechanical and vibrational behavior of the plates made of these materials. In order to pursue to this goal, a multi-scale approach has been considered. The properties of the carbon nanotubes are acquired using a series of molecular dynamics simulations. Subsequently, the derived stiffness matrix is used in micromechanics formulations to attain the homogenized elastic properties. The results are also compared with tensile experiments conducted on the nanocomposite tensile bars. Finally, the vibration behavior of the nanocomposite plates were attained using an in-house finite element code. The influence of content and arrangement of the nanotubes on the mechanical and vibrational behavior of the nanocomposites were investigated in detail.

2 EXPERIMENTS

The multi walled carbon nanotubes used in this study are catalytic chemical vapor deposition produced thin MWCNTs (NC 7000™) by Nanocyl SA, Belgium, with the average aspect ratio of 67 ($d_{ave}=10.4 \text{ nm}$), and 7-9 walls [7]. Nanocomposites containing different contents of the aforementioned MWCNTs namely, 0.5, 1.0, 3.0, and 5.0 wt. % were prepared using a conical counter rotating twin-screw extruder (HAAKE™ Rheomex CTW, $\Phi=31.8/20 \text{ mm rear/front, } L=300 \text{ mm}$). Subsequently, injection molding of the
nanocomposite specimens was performed on Ferromatik, Milacron following the instructions of ISO 294-1 standard. The geometry of the cavity of the mold was dog-bone shape which was designed based on ISO 527-2 2012 standard. A series of experiments were conducted using a two level, four factor factorial design to investigate the influence of the four considered parameters on the mechanical properties of the nanocomposites. In order to characterize the mechanical properties of the injection-molded specimens, uniaxial tensile experiments were conducted according to ISO 527-1 2012 standard. From each series of specimens produced via the defined setting and content, 10 samples were selected randomly, and tested in the dry as molded state.

3 MULTISCALE MODELING

A multiscale approach is devised in the three consecutive steps. Firstly, MD simulations were employed to determine the transversely isotropic stiffness tensor of the multi-walled carbon nanotubes. In the second step, acquiring the nanofillers (reinforcement) and polymer (matrix) stiffness matrices, the effective elastic moduli of the nanocomposite are scaled up from the nanoscale to the microscale using the Benveniste-Mori-Tanaka method. Finally, the homogenized stiffness matrixes of the nanocomposites based on the fillers’ arrangements are incorporated in the finite element code to attain the vibrational behavior of the thin plates.

3.1 MOLECULAR DYNAMICS

Molecular dynamics simulations were conducted to determine the elements of the stiffness matrix of the multi-walled carbon nanotubes at the atomistic scale. MD simulations offer an appropriate and effective means to deal with large systems and relatively longer simulation times. All MD simulations runs were conducted with large-scale atomic/molecular massively parallel simulator (LAMMPS) [8]. For the short-range bonding atomistic interactions within each layer of the multi-walled carbon nanotubes, Adaptive Inter-molecular Reactive Empirical Bond Order (AIREBO) potential, which is a multi-body force field developed for hydro-carbons was considered [9]. The AIREBO potential includes the second generation REBO potential $E^\text{REBO}$ [10], the torsion potential $E^\text{TORSION}$, and the Lennard Jones potential $E^\text{LJ}$:

$$E^\text{AIREBO} = \frac{1}{2} \sum_{i} \sum_{j 
eq i} \left[ E^\text{REBO}_{ij} + E^\text{LJ}_{ij} + \sum_{k 
eq i} \sum_{l 
eq j,k} E^\text{TORSION}_{ijkl} \right]$$

The second generation REBO potential $E^\text{REBO}$ controls the covalent bond interactions. In addition, $E^\text{TORSION}$ and $E^\text{LJ}$ control dihedral rotation of bonds, and long range non-bonded interactions between the atoms within each single layer, respectively. The REBO potential is expressed as:

$$E^\text{REBO}_{ij} = V_R(r_{ij}) - b_{ij}V_A(r_{ij})$$

where $V_R$ and $V_A$ represent the interatomistic repulsion and attraction terms, respectively. Moreover, $r_{ij}$ is the distance between the neighboring atoms $i$ and $j$, and $b_{ij}$ is the reactive empirical bond order. This force field has been successfully used in other works to simulate the mechanical behavior of carbon nanotubes and Graphene layers [11,12]. In order to include the non-bonded Van der Waals (VdW) interactions between the Carbon atoms in the different layers (walls) of the multi-walled tubes, long-range LJ 12-6 potential was added to
the total energy:
\[ E^{\text{tot}} = 4\varepsilon \left[ \left( \frac{\sigma}{r} \right)^{12} - \left( \frac{\sigma}{r} \right)^{6} \right] \]
where \( \varepsilon \) is the well-depth energy, and \( \sigma \) is the equilibrium distance [13].

Conjugate gradient algorithm was used to minimize the total potential energy of the initial configurations. The equations of motion in all MD simulations were integrated using Velocity Verlet algorithm with integration time step of 1 fs at zero temperature. It is also noteworthy to mention that the shrink-wrapped boundary conditions were imposed on all directions of the simulation box.

In order to attain the stiffness elements of the transversely isotropic nanotubes, a method based on the determination of the elastic energy as the function of the applied strain was considered [9]. In this method, the total potential energy of the system is calculated and recorded as the function of the imposed strains. Therefore, in different loading scenarios, the values of the corresponding stiffness in the system (six elements in the transversely isotropic stiffness matrix) are calculated:
\[ C_{ijmn} = \frac{1}{V} \frac{\partial^2 U}{\partial \varepsilon_m \partial \varepsilon_y} \]
where \( U \) is the potential energy in the system, and \( \varepsilon \) is the imposed strain on the carbon nanotubes. The volume of the single walled carbon nanotubes has also been calculated as:
\[ V = \pi D t L \]
where \( D \) and \( L \) are the diameter and length of the tube. Thickness of a single nanotube layer has been considered \( t = 3.4 \ \text{Å} \), same as the thickness of an individual graphene layer.

Figure 1 (a) and (b) show the influence of the nanotube diameter on the elastic and shear modulus of the Single Walled Carbon NanoTubes (SWCNTs). Single walled carbon nanotubes are carbonic cylinders comprising of a single layer of graphene, and their properties based on their chiral vector \( (C = (n, m)) \), which define their atomistic arrangement and geometry can be completely different. Armchair carbon nanotubes \( (C = (n, n)) \) do have a metallic and conductive structure, while Zigzag SWNTs \( (C = (n, 0)) \) are semiconducting, unless the corresponding chiral number is a multiple of three. Since the chiral vectors of each single layer in the multi-walled carbon nanotubes (MWCNTs) have not been characterized, both mentioned atomistic arrangements are simulated to study their elastic behavior in detail. As it can be seen from the Figure 1, while Zigzag nanotubes are showing higher longitudinal moduli of elasticity, they are responding less stiff in the torsion. Moreover, with the increase in the diameter of the nanotube, the elastic values for both of the types decrease, especially in the lower diameters where the plunge in the stiffness values are significant. However, with further increase in the diameter of the nanotubes higher than 2.0 nm, the rate of decrease in the elastic stiffness values reduces notably.
Figure 1: Influence of the nanotube diameter on the (a) longitudinal elastic modulus along the major axis of the tube, and (b) shear modulus of the SWCNTs.

Figure 2 shows how the longitudinal and shear stiffness of the multiwalled carbon nanotubes change with the increase in the number of walls. As it is obvious from the presented curves, increasing the number of the tubes leads to higher stiffness values. Moreover, this increase in the stiffness of the MWCNTs as the function of the wall numbers is consistent. However, it should also be mentioned that in the simulations, loading on the entire boundary (i.e. all layers) was considered. This scenario of loading is not realistic e.g. for shear response, where the load is carried by the outermost layer, and afterwards stresses and strains are transferred to the inner layers through VdW interactions. If the outermost layer of the multi walled tubes carries the load, one should expect that the presented values drop dramatically.

Figure 2: Influence of the number of the walls on the elastic modulus and shear modulus of MWCNTs

3.2 Micromechanics model

The acquired the stiffness matrix of the nanofillers from MD simulations along with the isotropic elastic properties of the neat polymer (PA 66) is used as input to the micromechanical model to estimate the bulk elastic modulus of the nanocomposites including uniformly dispersed and straight carbon nanotubes. Using the equivalent continuum model based on the Eshelby–Mori–Tanaka approach enables us to attain the homogenized stiffness matrix of the nanocomposites [14,15]. The effective stiffness tensor $[C]$ of the two-phase
nanocomposites can be estimated as [16]:

\[
C = C_m + c_r \left( (C_r - C_m) A_r \right) \left( c_m I + c_r A_r \right)^{-1}
\]

(6)

where \( C \), \( C_m \), and \( C_r \) are nanocomposite, matrix, and reinforcement stiffness tensors, respectively. In addition, \( c_m \) and \( c_r \) are matrix and reinforcement volume fractions, and \( I \) is the identity tensor. \( A_r \) represents the dilute mechanical strain concentration tensor:

\[
A_r = \left[ I + S (C_m)^{-1} (C_r - C_m) \right]^{-1}
\]

(7)

where \( S \) is Eshelby tensor that in case of cylindrical inclusions like the used carbon nanotubes, it can be estimated based on the Mura’s theory [17]:

\[
S_{nm} = \begin{bmatrix}
5 - 4v_m & 4v_m - 1 & v_m \\
8(1 - v_m) & 8(1 - v_m) & 2(1 - v_m) \\
4v_m - 1 & 5 - 4v_m & v_m \\
8(1 - v_m) & 8(1 - v_m) & 2(1 - v_m)
\end{bmatrix}
\]

(8)

Acquiring the global stiffness matrix in the defined principal direction, global stiffness matrix of the nanocomposites comprising the carbon nanotubes that are oriented in other directions can also be obtained using:

\[
C^* = N_{\phi}^T C N_{\phi}
\]

(9)

Additionally, the homogenized elastic isotropic properties of the nanocomposites containing fully random nanofillers can also be accessed. Stiffness matrix for the \( r^{th} \) phase as a function of transversely isotropic stiffness matrix and Euler angles \( (\theta, \phi) \) can be calculated as:

\[
C^* = N_{\theta}^T N_{\phi}^T C N_{\phi} N_{\theta}
\]

(10)

where \( N_{\theta} \) and \( N_{\phi} \) are transformation matrixes along Euler angles \( \theta \) and \( \phi \), respectively. Therefore, the integration of the stiffness matrix for the \( r^{th} \) phase in the span of \([-\pi, +\pi]\) results in the global matrix of a nanocomposite comprising reinforcements in all direction in the three dimensions. Integration of the stiffness matrix for the \( r^{th} \) phase in the span of \([-\alpha, +\alpha]\) would also create a scenario like what is happening during the process of the injection molding:

\[
Q^* = \int_{-\pi}^{\pi} \int_0^\alpha C^*(\theta, \alpha) \frac{\sin \alpha}{2\pi (1 - \cos(\alpha))} d\alpha d\theta
\]

(11)

Figure 3 shows different possible arrangement scenarios of the nanofillers within the
polymeric matrix based on the design and manufacturing process. In fact, in case of thermoplastic-based nanocomposites reinforced with short fiber/nanofillers, design parameters like gate position, type of gate, thickness and geometry of the component, and processing parameters like viscosity of the melt, melt temperature, injection speed, etc. define the final arrangement of the fillers in the matrix.

Figure 3: Schematic of the nano-reinforcements arrangements in the nanocomposite: (a) aligned in the direction of loading, (b) aligned but not in the direction of loading, (c) some level of alignment, and (d) fully random.

Figure 4(a) shows the comparison between the predictions of the presented micromechanics models and the experimental results for the longitudinal elastic moduli. As it can be seen from the results, until 1.0 wt. % inclusion of the MWCNTs in the nanocomposite system, results lie in between the fully aligned and random predictions, and mostly toward the random prediction. However, in higher contents of the MWCNTs, the experimental results follow the trend of the fully random predictions, while being slightly less than those values. Moreover, the experimental and the presented model results are compared with the well-known Halpin-Tsai model, and its modified model for carbon nanotubes (Figure 4(b)) [18,19]. All the models are considering a fully random arrangement of the fillers in the polymeric matrix. As it can be noted, the original Halpin-Tsai and the modified Halpin-Tsai tend to overestimate and underestimate the elastic moduli, respectively. However, the presented model lie in between those predictions. It should be noted that some influential parameters such as curvature and defects of the tubes, their entanglements and agglomerations are neglected in the mathematical predictions. In addition, the applied loading scenarios in the molecular dynamics were loading to all layers, which is assumed to heighten the observed overestimation in the introduced model.

Figure 4: Comparison between experimental results and (a) the predictions of the presented micromechanics model, and (b) the predictions of the different models for the elastic moduli of the composites.
3.3 Finite element of plate vibration

In order to investigate the vibration of the nanocomposite plates, an in-house finite element code based on the Mindlin-Reissner plate theory was employed [20,21]. In this regard, it is assumed that strain in mid-surface plane of the thin plate is zero, and straight lines that are normal to the surface remain straight. Therefore, the displacement vector is considered in the following form:

\[
\begin{bmatrix}
U(x, y, z, t) \\
v(x, y, z, t) \\
w(x, y, z, t)
\end{bmatrix} =
\begin{bmatrix}
u(x, y, z, t) \\
v(x, y, z, t) \\
w(x, y, z, t)
\end{bmatrix}
\]

(12)

where \(w\) as the out-of-plane displacement, in addition to the two rotational angles \(\psi_x\) and \(\psi_y\) are used to describe the displacement of the plate. It is also noteworthy to mention that the out-of-plane displacement is assumed to be independent of \(z\).

Strain vector for the plate is also defined as:

\[
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy} \\
\gamma_{xz} \\
\gamma_{yz}
\end{bmatrix} =
\begin{bmatrix}
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \\
\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} + \frac{\partial w}{\partial z} \\
\frac{\partial w}{\partial x} + \frac{\partial u}{\partial y} + \frac{\partial v}{\partial z} \\
\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} + \frac{\partial u}{\partial x} \\
\frac{\partial w}{\partial z} + \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}
\end{bmatrix}
\]

(13)

Considering the plate thin enough, plane stress assumption is made. Therefore, the stress-strain relation for the nanocomposite plate comprising uniformly aligned nanofillers will be:

\[
\begin{bmatrix}
\sigma_{11} \\
\sigma_{22} \\
\sigma_{23} \\
\sigma_{31} \\
\sigma_{12}
\end{bmatrix} =
\begin{bmatrix}
D_{11} & D_{12} & 0 & 0 & 0 \\
D_{12} & D_{22} & 0 & 0 & 0 \\
0 & 0 & D_{44} & 0 & 0 \\
0 & 0 & 0 & D_{55} & 0 \\
0 & 0 & 0 & 0 & D_{66}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_{11} \\
\varepsilon_{22} \\
\varepsilon_{23} \\
\varepsilon_{31} \\
\varepsilon_{12}
\end{bmatrix}
\]

(14)

As the result of the equilibrium conditions, the equations of motion will be:
\[ \rho \ddot{h} = \frac{\partial Q}{\partial x} + \frac{\partial Q_y}{\partial y} - p(x, y, t) \]

\[ \frac{\rho h^3}{12} \ddot{y}_x = Q_y - \frac{\partial M_x}{\partial y} - \frac{\partial M_{xy}}{\partial y} \]

\[ \frac{\rho h^3}{12} \ddot{y}_y = Q_y - \frac{\partial M_x}{\partial x} - \frac{\partial M_{xy}}{\partial x} \]

where shear \((Q)\), and moment \((M)\) resultants are:

\[
\begin{bmatrix}
Q_1 \\
Q_2
\end{bmatrix} = k \int_{-\theta}^{\theta} \begin{bmatrix}
D_{55} & 0 \\
0 & D_{44}
\end{bmatrix} \begin{bmatrix}
\varepsilon_{31} \\
\varepsilon_{32}
\end{bmatrix} d\theta
\]

\[
\begin{bmatrix}
M_{11} \\
M_{22} \\
M_{12}
\end{bmatrix} = \int_{-\theta}^{\theta} \begin{bmatrix}
D_{11} & D_{12} & 0 \\
D_{12} & D_{22} & 0 \\
0 & 0 & D_{66}
\end{bmatrix} \begin{bmatrix}
\varepsilon_{11} \\
\varepsilon_{22} \\
\varepsilon_{12}
\end{bmatrix} d\theta
\]

where \(k\) is the shear correction factor.

The presented governing equations were incorporated in the finite element code using 4-node elements to characterize the vibrational behavior of the nanocomposite plates. The presented results reflect the properties of a thin plate with the dimensions of \(80\times40\times0.4\) cm.

Figure 5 (a) and (b) show the influence of the level of alignment (Case c in Figure 3) of carbon nanotubes on the first and second vibration modes. In the figures the left end of the scope \((\theta = 0)\) represents the fully aligned nanofillers in the nanocomposite system, and the right end of the angular scope \((\theta = \pi/2)\) represents a fully random arrangement of the tubes in the composite. Figure 6 is also showing the influence of MWCNT content in the polymeric system on the vibration modes while the tubes are aligned and random. The results show how the level of alignment in combination of the content can change the vibration behavior significantly.

![Figure 5: The influence of the level of nanotubes alignment in the composite system on the (a) first, and (b) second mode of vibration](image-url)
Figure 5: The influence of the nanotube content and arrangement in the composite system on the (a) first, and (b) second mode of vibration

Figure 6 and 7 show the absolute amplitude response of the nanocomposite plates in their first three modes of excitation. As it can be seen from the results, in the nanocomposite plates comprising fully aligned nanotubes, first excitation mode happens in lower frequency. In fact, while a fully random nanocomposite plate is isotropic in all directions, the transverse isotropic properties in the aligned case are resulting in the observed response. Moreover, with the increase in the content of the nanotubes in the composite system, the excitation frequencies are happening in higher frequencies. In fact, with the increase in the content of the nanotubes, the stiffness values of the composite plates in all directions increase. Therefore, the observed decrease in the amplitude values is expected in the vibration behavior. It should be also mentioned that in the simulations, the influence of the nanotube arrangement and shear-lag parameter on the damping of the plate is not considered. The damping in all the different presented nanocomposite plates is considered to be the same as the neat polymer.

Figure 6: The absolute amplitude with excitation frequencies in nanocomposites comprising fully aligned and random arrangement of the nanofillers.
4 CONCLUSIONS

A multiscale approach was pursued to investigate the mechanical and vibration behavior of the nanocomposite thin plates. Different levels of this approach namely Molecular Dynamics, micromechanics, and finite element method provided a comprehensive understanding on how the arrangement and content of the nanotubes influence the different behaviors of the nanocomposites. While molecular dynamics simulation results show the influence of the parameters such as diameter and number of layers in the tubes on the different properties of the carbon nanotubes, micromechanics modeling provides a comprehensive insight on the impact of the design and processing on the bulk properties of the nanocomposites. Finite element simulations also provide the understanding on the response of the thin plates made of the nanocomposites with the different microstructures and contents. The presented multiscale approach provides an insight for designing and tailoring the carbon nanotubes reinforced nanocomposite thin plates for the desired mechanical and vibrational applications.

REFERENCES


MOULD-IN MANUFACTURING STUDY OF THERMOPLASTIC COMPOSITES WITH NOVEL INTERLOCKING SURFACE STRUCTURES FOR ADHESIVELY BONDED JOINTS

John Lyons*, Ronan M. O’Higgins*, Conor T. M’Carthy*

*Bernal Institute
University of Limerick, Limerick, Ireland
john.lyons@ul.ie

Key words: Fibre Reinforced Composites (FRPs), Mould-in Manufacturing, Fibre Volume Fraction (FVF)

Summary: The aim of the current research is to manufacture novel structured surface topology for composite-to-metal adhesively bonded joints that will mechanically interlock in shear. Mould-in manufacturing of textile fibre reinforced plastics (FRPs) has been identified as a method to cast the interlocking designs on the composite adherend. Mould-in manufacturing offers many advantages over other industry subtractive machining methods such as maintaining fibre continuity, an increased fibre volume fraction (FVF) around the mould-in structure and the method causes no damage to the final composite structure. The present work investigates the effects of mould-in manufacturing on the composite structure morphology and FVF distribution around interlocking profiles. Mould-in manufacturing was successfully employed to cast profile designs on composite adherends and there was no issue with interlocking profiles fitting with metal adherends.

1 INTRODUCTION

The advantages of high strength- and stiffness-to-weight ratios make fibre reinforced plastics (FRP), an ideal candidate for primary automotive and aerospace structures where weight and fuel efficiency are important considerations [1]. Near-net shape manufacturing is another advantage of FRPs, which reduces the number of overall components compared to metallic materials. However, despite these advantages the requirement for joining of FRPs to other structural materials cannot be eliminated. The main joining methods employed are mechanical fastening (bolts, rivets and screws), adhesively bonded joints and hybrid mechanically fastened/adhesively bonded joints [2]. Mechanical fasteners have the advantages of being reliable compared to other joining methods and permit disassembly for repairs and end of service/life disposal. However, fasteners add weight to the structure, requiring post-manufacturing drilling and assembly [3, 4]. The fastener holes introduce highly localised stress concentrations and interrupting continuity of fibres [4] weakening the laminate and reducing overall efficiency of the structure. Adhesively bonded joints distribute load over a larger interfacial area compared to mechanical fasteners, and also reduce the additional weight and costs compared to fasteners [4]. However, bonded joints require surface treatment of adherends and are susceptible to changes in environmental conditions, while low interlaminar shear and tensile strength limit joint efficiency [5].

The development of composite–to-metal adhesively bonded joints with novel structured surface topology on the bondline that will mechanically interlock in shear is presented herein. Manufacturing of the interlocking profiles on composite adherends
presents some interesting challenges. One concept is mould-in manufacturing that allows for casting of the interlocking profiles during the manufacturing process. The preform fabric yarns are rearranged around structures in the tool moulding or mould inserts to cast the required shape in the composite adherend structure. Figure 1 illustrates the mould-in manufacturing concept.

Previous authors have investigated mould-in manufacturing as alternative for drilling of holes in composite panels [6, 7, 8]. The mould-in holes maintain the continuity of the fibres and eliminate the potential for matrix micro-cracking and delamination damage in composite structure from drilling [6]. A study of mould-in hole bearing strength [7] found that the increased number of 90° tows in the hole region helped increase the bearing strength. It was shown by [6] that in order to fully understand the effects of such mould-in features on the composite structure, the local variation of fibre volume fraction (FVF) around the feature should be determined. Various methods to characterise local fibre volume fraction at mould-in holes region were developed by [8], using a 2/2 twill commingled glass fibre polypropylene material. It was found the all methods investigated show similar results for FVF. However, it was shown that micrographs provided the best resolution compared to other methods, such as computer tomography and ignition loss methods.

The aim of the current research is to investigate the potential of mould-in manufacturing method for casting of interlocking structured surface topology designs in composite adherends for use in hybrid composite-metal joining applications. The effects of the mould-in casting on composite morphology and FVF around the casted regions were investigated. Dimensional metrology was performed on the resulting profiles to determine the effect of cure shrinkage and thermal contraction of the composite constituents on the geometry of the resulting profiles.

2 EXPERIMENTAL METHODOLOGY AND MATERIAL PROCESSING

For this investigation micro lap shear specimens were manufactured that allow the study of single profile designs in isolation. It also allows for in-situ testing within the chamber of an SEM [9] enabling the investigation of deformation and damage mechanisms occurring around a single interlocking profile. Three different surface topology profiles were moulded to determine the fibre volume fraction (FVF) distribution and morphology of the composite structure. Figure 2 shows the structured surface topology profile designs manufactured for the investigation. Configuration 1 (Figure 2 (a)) employs interlocking profile design which is long in the loading direction of the micro lap shear specimen but
narrow in the width direction. Conversely, the design for configuration 2 (Figure 2 (b)) is narrow in the loading direction but long in the width direction of the joint. Configuration 3 (Figure 2 (c)) is a square interlocking profile design. All three profiles were designed to fit in the centre of the 7mm x 5mm bond area of the micro-lap shear specimen.

The material used is a 2/2 twill commingled fabric (Schappe Techniques), the carbon fibre is a stretch broken fibre (SBF) and is co-wrapped in a Polyamide 12 (PA 12) filaments. The average length of the stretch broken fibre is 80 mm, but some can be up to 200 mm in length. The reinforcing effect of SBF is as effective as that of continuous fibres [10]. Each fabric ply gives a consolidated theoretical thickness of 0.27 mm, 8 plies of fabric were cured to give a laminate thickness of roughly 2.16 mm. Preforming is the key process step for mould-in manufacturing, the rearranging of the fabric yarns around the moulding structures is achieved at this stage. The preform fabric thickness before curing is roughly 1.2 mm thick and the mould-in profiles are 0.75 mm in depth, the theoretical ply consolidate thickness is 0.27 mm. This means that the fabric yarns need to be arranged around the mould geometry for the first 4 plies of the laminate. Each of the first 4 plies were placed on the aluminium tool plate individually and the yarns rearranged around the moulding structures. After this these plies were combined and the remaining fabric plies where added to complete the preform. The preform was vacuum bagged on an aluminium tool plate and cured in an autoclave at a temperature of 220 °C for 20 mins with 6 Bar pressure applied. Specimens were extracted using a diamond blade cutting saw, with the overall micro-lap shear dimensions being 7 mm x 24.5 mm. Each profile was located on the centre of the bonded area (7mm x 5mm).

Micrographs of the samples were captured to see the effects of mould-in manufacturing on the composite structure morphology, and to measure the local FVF around the structured surface topology profiles. Figure 3 (a) shows the section planes taken for analysis of each configuration, these were in the loading direction and the width direction (Figure 3 (b) & (d)). The specimens where mounted in clear epoxy resin and ground to the plane of interest. A course silicon carbide (SiC) paper (P240) was used to grind samples to plane of interest and the SiC papers of P600 and P1200 were used to more finely grind the
samples before polishing. To achieve the polished surface for the micrographs, mounted samples were polished down to a final particle size of 0.05 micron. An Olympus BX60 optical microscope was used to capture the micrographs of the composite adherend with structured surface topology profiles using magnifications of 5x and 10x. In order to capture all the composite structure, images had to be systemically taken at resolutions of 0.954 µm and 0.477 µm using 5x and 10x objectives, respectively. Micrograph images of the samples were used for the metrology analysis, which measured depth (D), angle (θ) and radii (r) of the profiles as highlighted in Figure 3 (c).

3 RESULTS

3.1 Metrology Analysis

The geometry of the resulting mould-in profiles for each configuration where measured to determine the important feature dimensions. A spline was drawn on the micrographs that followed the profile shape for both the width and loading direction. This was exported as x-y coordinates into Matlab where each feature was then isolated and fits were applied to the data to determine feature dimensions. Table 1 shows the mean depth, angle and radii measurements for each profile configuration compared to the target dimension. Depth measurements range between 97 - 99% of the target dimension of 750 µm. The angle measurements were also very close to the target dimension, being only 1-3° off the target dimensions of 85°. These measurements show the minimal effects of thermal and curing properties of the composite constituents on the casted profiles dimensions.
Table 1: Summary of metrology results

<table>
<thead>
<tr>
<th>Specimen Type</th>
<th>Measurement Type</th>
<th>Target dimension</th>
<th>Mean</th>
<th>Standard deviation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Configuration 1</strong></td>
<td>Depth (μm)</td>
<td>750</td>
<td>731</td>
<td>± 15</td>
</tr>
<tr>
<td></td>
<td>Angle (°)</td>
<td>85</td>
<td>82</td>
<td>± 2.61</td>
</tr>
<tr>
<td></td>
<td>Radius (μm)</td>
<td>250</td>
<td>263</td>
<td>± 81</td>
</tr>
<tr>
<td><strong>Configuration 2</strong></td>
<td>Depth (μm)</td>
<td>750</td>
<td>733</td>
<td>± 17</td>
</tr>
<tr>
<td></td>
<td>Angle (°)</td>
<td>85</td>
<td>84</td>
<td>± 1</td>
</tr>
<tr>
<td></td>
<td>Radius (μm)</td>
<td>250</td>
<td>287</td>
<td>± 100</td>
</tr>
<tr>
<td><strong>Configuration 3</strong></td>
<td>Depth (μm)</td>
<td>750</td>
<td>740</td>
<td>± 4.4</td>
</tr>
<tr>
<td></td>
<td>Angle (°)</td>
<td>85</td>
<td>83</td>
<td>± 1.5</td>
</tr>
<tr>
<td></td>
<td>Radius (μm)</td>
<td>250</td>
<td>306</td>
<td>± 113</td>
</tr>
</tbody>
</table>

The radii measurements for each profile design showed a larger standard deviation compared to the depth and angle measurements. The mean measured radii were 5 - 22.4% larger that the target dimension of 250 μm. Table 2 below shows the radii measurements, separated based on location i.e. top or bottom of the structured surface topology profiles. From Table 2, radius location is the why there was a large variation in measured values. The top radii are concave leading to a reduction in radius, while the bottom radii are convex increasing the radii. This is due to the tool-part interaction and curing of the material. Modelling of the process will help to explain the tool and part interaction at these radii regions.

Table 2: Radii measurements by location on structured surface profiles

<table>
<thead>
<tr>
<th>Specimen Type</th>
<th>Top Radii (μm)</th>
<th>Bottom radii (μm)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Configuration 1</strong></td>
<td>191</td>
<td>336</td>
</tr>
<tr>
<td><strong>Configuration 2</strong></td>
<td>206</td>
<td>368</td>
</tr>
<tr>
<td><strong>Configuration 3</strong></td>
<td>206</td>
<td>407</td>
</tr>
</tbody>
</table>

Figure 4 shows the micrograph of the composite structure with a superimposed metal adherend profile. This micrograph shows configuration 2 across the width direction, it is clear that the composite adherends provide a good fit for the male metal adherends, the dimensional tolerances of which are much more easily controlled through the milling machining process. The bondline thickness is consistent at the bottom of the profiles shape. There is an observable increase in the bondline thickness around the top radii regions due to the top radii being smaller and the resulting angle being slightly under 85°. The effect of thermal and curing properties of the composite along with the thermal properties of the moulding material could be the cause of this variation in bondline thickness in the top surface region. However, this effect is relatively small and doesn’t prevent adherends fitting together.
3.2 Effects on Morphology

Figure 5 (a) shows the overall microstructure of the cured composite structure when no mould-in casted profile is introduced. The micrograph is looking in at the 0° (warp direction) and the 90° (weft direction) running parallel. There are no large macro size voids present indicating that good consolidation was achieved. Figure 5 (b) is an image for the composite structure at 50x magnification, showing the excellent consolidation quality achieved with autoclave processing.

Figure 6 shows the composite microstructure with a mould-in structure surface profile. This is the square profile design (configuration 3) sectioned across the width direction, as highlighted in Figure 6 (a). Figure 6 (b) shows the overall composite structure, where there is a reduction in the 90° tows present due to the movement of the fibre yarns around the moulded structure. In Figure 6 (c) at 20x magnification the 90° tow can be seen in more detail. These fibre shapes are not elongated oval shapes, which are typical for 90° fibres, indicating that these fibres are not aligned perpendicular to the sectioned plane and are curving to be moulded around the profile shape. There is a large resin rich region present near the two top radii regions of the profile; this was also observed on other specimen
micrographs. This issue was not seen in bottom radii region of the casted samples across all the micrographs of the specimens. The bottom radii regions are a more open angle, 275° for the fibres to be moulded onto compared to the top radii regions which is only a 95°.

Figure 6: Configuration 3 morphology (a) section plane, (b) 5x image of composite morphology (c) 10x image of composite morphology

Figure 7 shows the loading direction of the configuration 2, where the section plane of interest is highlighted in Figure 7 (a). Figure 7 (b) shows the overall structure, when compared to the configuration 3 profile of Figure 6, there is no large resin rich regions observed in top radii regions. Figure 7 (c) shows specimen at 20x magnification and how the fibre tows moulded to the bottom region of profile shape, this was observed across all samples. Figure 7 (b) region (I) shows increase in 90° fibres which will help increase the damage tolerance of the structure similar what was report by [7], who concluded that an increase in fibres perpendicular to loading direction after the mould-in holes helped to increase bearing strength.
3.3 Effects on Fibre Volume Fraction

The micrographs at 20x magnification were reconstructed to show the full composite structure as shown in Figure 8 (a) for each profile design in width and loading directions. In order to carry out an in-depth analysis of the local FVF around the mould-in profiles the micrograph were converted to grey scale and binarized by thresholding as shown in Figure 8 (b) & (c). The micrograph image dimensions for all samples were approximately 7 mm x 2.2 mm. For the FVF analysis, this image was split into a 24×8 grid as shown in Figure 8 (a), where the square size of each grid corresponded to the approximate thickness of one ply. The FVF was measured for each area of the grid to determine the FVF variation across the image and through the thickness. To help interpret how the FVF varied across the micrographs an interpolation function was applied using Gwyddion scanning probe microscopy analysis software that can be used for data visualization [11].
Figure 8: FVF analysis methodology (a) structured grid, (b) threshold of image (10x) & (c) binarized image (10x)

Figure 9 shows the FVF measurements across the composite structure without any mould-in profile. The sample was 6.5 mm in length and roughly 2 mm thick. Figure 9 (a) shows the results with the interpolation function applied to the FVF results. Figure 9 (b) shows the average FVFs across the width, while Figure 9 (c) shows the average FVFs measured through the thickness. The shade region on each graph shows the manufacturers stated FVF of 52 ± 3%. The mean FVF across the sample area was 53% which was within the stated FVF by the manufacturer. In Figure 9 (c) it is clear to see that the FVF through the thickness is below average in the bottom region. The resin rich area in the bottom region would be reduced with higher consolidation pressure.

Figure 9: FVF analysis with no mould-in profile (a) interpolated FVF image, (b) average FVF across the width of specimen & (c) average FVF through the thickness of specimen
Figure 10 shows the effects of mould-in structure on the FVF. This structure was the square profile design (configuration 3), this micrograph is in the loading direction on the specimen. The same bar charts where employed to show how the average FVF varies across the width (Figure 10 (b)) and through the thickness (Figure 10 (c)) of the composite structure. A clear increase in the FVF is observed through the thickness below the mould-in profile, where the average FVF in this region below the mould-in profile is 61%, an increase of 8% over the measured mean FVF for the bulk material. Compared to the average FVF of 53% for composite sample with no mould-in region, an average was 59% was measured for the profile shown in Figure 10 (a). The micrographs across all samples didn’t show trend of fibre volume increase across the width of the specimen structures, only through the thickness of composites structure. This is due to the fibre yarns being able to move during both preforming (pressure application) and consolidation stages. Mould-in holes through the full thickness of structure from literature, constricted the fibres to just areas around the mould-in hole region and this gives more control of where higher FVF is located compared to the mould-in casting of structure profiles. This will help increase the damage resistance through the thickness of the composite adherends.

Table 3 shows the average FVF for each profile design in both the loading and the width directions. The configuration 1 FVF is within the tolerance stated by the manufacturer in both directions. Configurations 2 & 3 have higher FVF averages than the tolerances stated by manufacturer apart from Configuration 2 loading direction, showing how mould-in manufacturing is increasing the FVF in the profile surrounding area.
Table 3: Average FVF for each configuration in loading and width directions

<table>
<thead>
<tr>
<th>Specimen Type</th>
<th>Direction</th>
<th>Average FVF (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Configuration 1</td>
<td>Loading direction</td>
<td>48</td>
</tr>
<tr>
<td></td>
<td>Width direction</td>
<td>53</td>
</tr>
<tr>
<td>Configuration 2</td>
<td>Loading direction</td>
<td>54</td>
</tr>
<tr>
<td></td>
<td>Width direction</td>
<td>62</td>
</tr>
<tr>
<td>Configuration 3</td>
<td>Loading direction</td>
<td>59</td>
</tr>
<tr>
<td></td>
<td>Width direction</td>
<td>57</td>
</tr>
</tbody>
</table>

4 CONCLUSIONS

The mould-in manufacturing was successfully employed to cast the structures surface topology profiles into the composite structure. The metrology analysis showed that the joint fit can be achieved and the small effect of composite’s cure and thermal properties had on the profile final dimensions along with mould material thermal properties. The micrographs showed that good consolidation has been achieved from autoclave processing. However, there were resin rich regions observed at the top radii regions of the profiles, this due to the region angle (95°) being is more closed off compared to the bottom radii region (275°). Increase of profiles size and modifying the top radii region shape will have the potential to reduce the size of the resin region, if required. The FVF analysis showed that there was slight increase in FVF around the mould-in regions. The FVF analysis showed a clear increase in the FVF through the thickness of the composite structure moving away from profile designs. The increase in the FVF across the width was not as clear due to the tow movement during manufacturing. However, the increase in FVF in the around mould-in region which will give a more damage tolerant structure compared to post-manufacturing machining which interrupting the continuity of fibres. Joint performance and damage progression analysis will be investigated, this will help inform where design modification can be made to improve the composite morphology and overall performance for single profile designs. This analysis with assist in design of multi-profile lap shear adhesively bonded joints.

5 ACKNOWLEDGEMENTS

This publication has emanated from research conducted with the financial support of Science Foundation Ireland (SFI) under Grant Number 13/IA/1833.

REFERENCES


EXPANDING THE WEIGHT-STRENGTH-CURVES FOR UNSTIFFENED CFRP SHELL STRUCTURES

Pawel Lyssakow*, Linus Friedrich*, Kai-Uwe Schröder*

*Institute of Structural Mechanics and Lightweight Design, RWTH Aachen University, Germany
Wüllnerstrasse 7, 52062 Aachen
Pawel.Lyssakow@sla.rwth-aachen.de

Keywords: Weight-Strength-Curves, CFRP shell structures, Structural concepts.

Summary: Thin walled shell structures are prone to buckling when axial compressive load is applied. These structures can be designed stiffened or unstiffened, using isotropic or anisotropic materials. During the structural design phase, the engineer needs to choose a structural concept considering three crucial parameter: the load carrying capacity, the structural mass and the costs of production. In order to compare the structural mass and the load carrying capacity of unstiffened isotropic and anisotropic shell structures this paper focus on expanding the Weight-Strength Curves (WSC) for unstiffened Carbon Fiber Reinforced Polymer (CFRP) shell structures. Different options to expand the WSC are introduced and evaluated in this paper. Concluding, the imperfection sensitivity of selected shell structures is studied.

1. INTRODUCTION

Thin walled shell structures are widely used for aerospace applications, as primary structures of aeroplanes or space launch vehicles. These structures are prone to buckling when axial compressive load is applied and are designed as stiffened or unstiffened shell structures. The determination of the bucking load and thus the load carrying capacity, as well as the knowledge of the mass of the structure is essential already during the preliminary design process. Shell structures can be designed as imperfection sensitive or imperfection tolerant. The crucial design parameter for imperfection sensitive structures, like unstiffened cylinders, are geometric imperfections of the structure. Geometric imperfections, that is small deviations of the real structure from the theoretically perfect shell structure, lead to a tremendous decrease of load

![Figure 1. Structural shell concept [1]](image-url)
carrying capacity. On the other hand, frame stringer stiffened shell structures are imperfection tolerant since the post-buckling regime of the skin fields is exploited. Various stiffened and unstiffened, shell concepts are shown in Figure 1. The structural design engineer is responsible to identify the structural shell concept that is most suitable for the given application at minimum total cost. In order to compare different structural concepts from a structural point of view and to allow for quick design decisions for or against a structural concept during an early design phase, Öry [2] derived in the 1990s the so called Weight-Strength-Curves, see Figure 2. On the ordinate the ratio of the equivalent thickness and the Radius of the shell structure $h/R$ is plotted. The average thickness $h$ is used since the thickness of stiffened shell structures is not constant among the shell surface. On the abscissa the ratio of the axial load per length and the Radius multiplied by the Young’s modulus $N_x/(RE)$ is represented. In Figure 2 the WSC are plotted for different structural concepts; for isotropic, isogrid and sandwich shell structures, as well as for corrugated shell structures having ring stiffeners on the inside and outside of the shell. For isotropic unstiffened shell structures, the imperfection sensitivity were considered according to Almroth using his 99% probability values [3]. The WSC reveal that the light weight design potential of shell structures is higher for stiffened than for unstiffened structures. For example it can be seen that the corrugated shell with ring stiffeners perform beneficial in comparison to the unstiffened shell.
Nowadays, an increased use in CFRP materials can be observed for aerospace structures, as i.e. for the Airbus A350 XWB [4]. These materials were not considered during the development of the WSC in the 1990s. In order to compare different structural concepts considering materials such as CFRP this paper aims to expand the WSC for unstiffened CFRP shell structures. In this paper, the WSC are expanded for unstiffened CFRP shell structures considering geometric as well as layup imperfections. Furthermore, the impact of geometric imperfections on the load carrying capacity of selected shell structures is studied.

2. EXPANDING THE WSC FOR UNSTIFFENED CFRP SHELL STRUCTURES

In order to expand the WSC for unstiffened CFRP shell structures a series of structural mechanical analyses were performed. In the following the structural model, used for this analyses is introduced. Subsequently, the modified WSC are introduced in order to enable a reasonable comparison of isotropic and anisotropic unstiffened shell structures. Concluding, the imperfection sensitivity of selected shell structures is evaluated and the relation of the manufacturing tolerance and the imperfections sensitivity is pointed out.

2.1 Structural model

In the 1990s, Zimmermann [5] performed a series of structural mechanical optimizations in order to determine the geometrical perfect shell structure with the highest buckling load. Among other results, Zimmermann developed the Z17 shell, a shell with the highest buckling load for the perfect geometrical cylinder. Since unstiffened shell structures are imperfection sensitive these imperfections should already be considered during the optimization process. Friedrich [6] investigated the influence of the stacking sequence on the buckling load of geometric imperfect CFRP shell structures under axial compression. For this purpose, Rotational Symmetric Imperfections (RSI) were taken into account to allow a closed form analytical description of the structural problem. The FL5 shell, which was derived in [6], has the highest buckling load for a geometric imperfect structure with rotational symmetric imperfections. The nominal properties of the Z17 and FL5 cylinders are summarized in Table 1. It can be seen that the geometrical dimension and the material properties are the same for both cylinders. The only difference between these two structures is the laminate’s layup. The buckling load of the perfect Z17 and FL5 shell were determined numerically by Friedrich [6] using an explicit displacement controlled manner, see table 1. The shell structures defined by Zimmermann [5] are assembled by different number of angle plies \( n_p \) which are varied form two to five. An angle ply consists of two unidirectional (UD) plies with an orientation \( \pm \alpha \) having a ply thickness of \( t_{UD} = 0.125 \text{mm} \) each. For \( \alpha = 0^\circ \) the fibre direction coincides with the shell’s longitudinal axis, the x-axis, whereby for \( \alpha = 90^\circ \) the fibre orientation coincides with the circumferential direction of the shell, defined as the y-axis. The laminate stacking sequence is given in positive z-direction, that is the first angle ply given is defined as the most inner ply of the shell. Corresponding to the Z17 and FL5 shell the following nomenclature is used further for a different amount of angle plies: The Z - shells are the structures with a layup leading to the highest buckling load for
Table 1. Properties of the Z17- and FL5 shells

<table>
<thead>
<tr>
<th>Property</th>
<th>Nominal data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length, $L$</td>
<td>510 mm</td>
</tr>
<tr>
<td>Radius, $R$</td>
<td>250 mm</td>
</tr>
<tr>
<td>Layup - Z17</td>
<td>$[\pm 30^\circ/90^\circ_2/\pm 23^\circ/\pm 38^\circ/\pm 53^\circ]$</td>
</tr>
<tr>
<td>Layup - FL5</td>
<td>$[\pm 38^\circ/\pm 68^\circ/90^\circ_2/90^\circ_2/\pm 38^\circ]$</td>
</tr>
<tr>
<td>Ply thickness, $t$</td>
<td>0.125 mm</td>
</tr>
<tr>
<td>Ply longitudinal Young’s modulus, $E_{11}$</td>
<td>123,551 MPa</td>
</tr>
<tr>
<td>Ply transverse Young’s modulus, $E_{22}$</td>
<td>8,707.9 MPa</td>
</tr>
<tr>
<td>Ply shear modulus, $G_{12}$</td>
<td>5695 MPa</td>
</tr>
<tr>
<td>Poisson’s ratio, $\nu_{12}$</td>
<td>0.32</td>
</tr>
<tr>
<td>Density CFRP, $\rho$</td>
<td>1.56 g/cm$^3$</td>
</tr>
<tr>
<td>Critical load Z17, $F_{cr,perf,Z17}$</td>
<td>255.37 kN</td>
</tr>
<tr>
<td>Critical load FL5, $F_{cr,perf,FL5}$</td>
<td>257.39 kN</td>
</tr>
</tbody>
</table>

geometric perfect shell structures and the FL - shells are the structures with a layup leading to the highest buckling load for geometric imperfect shell structures with having rotational symmetric imperfections. In this paper, using the same method as Friedrich [6], the buckling load of the imperfect Z and imperfect FL shell structures were determined based on the approach according to Koiter and Tennyson. As part of the conducted studies geometric imperfections were considered in the range of $w_t = 0, 1...0, 5$. Where $w$ is the imperfection amplitude and $t$ the thickness of the shell structure. Additionally, different layup imperfections in the range of $\pm 1^\circ...\pm 5^\circ$ in $1^\circ$ steps were considered.

2.2 Modified WSC diagram

As introduced in Section 1, the WSC were developed for isotropic material. In order to enable a proper comparison of different shell structural concepts, it is necessary to take into account that the usage of different materials is associated with different properties, as density or Young’s modulus. In this paper the different densities are considered as follows: Two ordinate axes are used for the expanded WSC diagram. On the left side of the WSC the ratio of the thickness and the radius for the isotropic (here aluminium) shell structure is illustrated $h_{iso}/R$, while on the right side the above mentioned ratio for CFRP shell structures can be seen $h_{CFRP}/R$. Thus, shell structures with the same mass are illustrated on a horizontal line in the WSC which enables a fair comparison of different shell structures regarding their structural mass. In the modified WSC diagram (Figure 3) two graphs can be seen, one for the perfect isotropic shell structure and one for the geometric imperfect shell structure, considering the 99% probability values according to Almroth [3]. Exemplary the critical buckling load for $P_{crit}$ for the geometric perfect and imperfect shell structure for $h_{iso}/R = 2.8 \cdot 10^{-3}$ is given in Figure 3. In this diagram the Young’s modulus of aluminium: $E^* = E_{Alu}$ was chosen. In order to enable a proper
comparison of shell structures regarding their load carrying capacity it is additionally important to modify the x-axis. As mentioned in section 1, $N_x$ is a function of constant values and the buckling load of the shell structure. For the here investigated and compared shell structures the radius is constant, see Table 1. The Young’s modulus, however, is constant and direction independent only for isotropic materials. Since it is desired to compare isotropic structures with CFRP structures, it is necessary to consider that first, the effective Young’s modulus needs to be determined in longitudinal direction of the shell structure and second, this effective Young’s modulus is a function of the shell structural layup.

In Figure 4 the WSC, as introduced in Figure 3, are shown expanded for the FL - and Z - shells. On the right y-axis of the diagram the values for angle plies with $n_p = 2, 3, 4, 5$ are marked. In e.g. for $n_p = 2$ the ratio is $\frac{h_{CFRP}}{R} = 2 \cdot 10^{-3}$. In the diagram the values for the geometrically perfect and imperfect (exemplary for $w_0/t = 0.2$) Z - and FL - shell structures are illustrated. The following Young’s modulus is used for this diagram:

$$E^* = \begin{cases} E_{Alu} & \text{for aluminium} \\ E_{CFRP} = \sqrt{E_{11}E_{22}} & \text{for CFRP} \end{cases}$$

A further possibility is to use for the CFRP structures the axial Young’s modulus of the particular shell structure. In this case the Young’s modulus is a function of the angle ply and of its stiffness. It can be determined using the classical laminate theory according to NASA [7]. In Figure 5 the
Figure 4. Modified WSC diagram: $E^* = E_{Alu}$ for aluminium and $E^* = \sqrt{E_{11}E_{22}}$ for CFRP

Figure 5. Modified WSC diagram: with $E^* = E_{Alu}$ for aluminium and $E^* = E_{eff}$ for CFRP
WSC are plotted for:

\[ E^* = \begin{cases} 
E_{Alu} & \text{for aluminium} \\
E_{eff} & \text{for the effective Young’s modulus of a particular shell} 
\end{cases} \tag{2} \]

Exemplary, in both Figures, Figure 4 and 5, for \( n_p = 5 \) and the isotropic structures with the same mass the critical buckling loads \( P_{crit} \) were shown for the shell structures studied. The following findings were made from the diagrams above:

- For isotropic structures the load carrying capacities in both diagrams were the same since the Young’s modulus of aluminium did not change.
- The load carrying capacities of the Z- and FL- shells are different in Figures 4 and 5 due to the different Young’s modulus.

In order to enable a reasonable comparison the following condition should be fulfilled for any combination of two shell structures \( i \) and \( j \):

\[
\text{If } P_{crit,i} > P_{crit,j} \text{ then } \left( \frac{N_x}{RE^*} \right)_i > \left( \frac{N_x}{RE^*} \right)_j \tag{3}
\]

By taking the critical load \( P_{crit} \) into consideration, the following findings resulted from the diagrams above:

- If the constant Young’s modulus \( \sqrt{E_{11}E_{22}} \) (Equation 1, Figure 4) for the CFRP structures was used the CFRP shell structures were comparable among each other, but, were not comparable with isotropic structures. This can be seen e.g. by the big gap in the WSC for the Z17 with RSI (129.1 kN) and the perfect isotropic structure (127.4 kN) despite just \( \Delta P = 2 kN \) difference.
- If the axial Young’s modulus of the particular shell structure \( E_{eff} \) (Equation 2, Figure 5) was used the CFRP structures were neither comparable among each nor with the isotropic structures.

As mentioned in Section 1, the load carrying capacity is crucial for the design of shell structures. In order to compare the load carrying capacity among each other using the WSC it is necessary to normalize the load with the same Young’s modulus, independent of the material used. The following example enables a reasonable comparison of shell structures using \( E^* = E_{Alu} \) for all shell structures, independent of the material. It can be seen that in Figure 6 the critical load \( P_{crit} \) (Equation 3) increases with increasing load carrying capacity. Thus, a reasonable comparison of the shell structures is possible. Though, the material used is neglected while determining the load carrying capacity.
2.3 Geometric imperfection sensitivity of selected shell structures

The WSC, expanded for unstiffened CFRP shell structures, were introduced in the previous section. Now it is possible to compare different structural concepts among each other. As mentioned in Section 1 imperfections reduce the critical buckling load of unstiffened shell structures significantly. Using the WSC, the influence of geometric as well as laminate layup imperfections can be illustrated easily. In Figure 7, the influence of geometric imperfections (RSI) on the load carrying capacity of the FL5 and Z17 shell is illustrated. To provide a better overview the Z17 and FL5 structures are plotted about each other, even if they have the same structural mass. For this diagram the constant Young’s modulus \( \sqrt{E_{11}E_{22}} \) (Equation 1) is used for \( E^* \). Due to the constant Young’s modulus a comparison of the structures is possible. It can be seen that the load carrying capacity of the geometric perfect Z17 shell is higher than of the FL5 shell. However, the load carrying capacity of the imperfect shell is higher for the FL5 shell, independent of the imperfection amplitude. Depending on the allowable imperfection amplitude the load carrying capacity is affected differently. Thus, the imperfection sensitivity of the shell structure depends on the manufacturing tolerance.
3. CONCLUSION AND OUTLOOK

In order to compare different structural shell concepts regarding their load carrying capacity and their structural mass the WSC were derived by Öry and expanded for unstiffened CFRP structures in this paper. In contrast to isotropic structures the load carrying capacity in the WSC depends on the layup of the composite structure. Therefore, for comparison reasons the Young’s modulus is decisive. In this paper the composite shells that were optimized for a geometric perfect and imperfect structure for \( n_p = 2, 3, 4, 5 \) are illustrated in the WSC. Different opportunities to expand the WSC in order to compare isotropic with anisotropic structures were discussed. However, this methods were not fully satisfying due to the dependency between the Young’s modulus and the critical buckling load. Using the same Young’s modulus for all structures, independent of the material leads to a reasonable comparison. Nevertheless, to compare different structural concepts this method appears unreasonable since the material used is neglected while determining the load carrying capacity. Therefore, it is desired to develop a shell structures specific parameter, that enables a fair comparison of isotropic and anisotropic shell structures, considering their structural mass as well as their load carrying capacity.

References


PERIDYNAMICS: CONVERGENCE & INFLUENCE OF PROBABILISTIC MATERIAL DISTRIBUTION ON CRACK INITIATION

Martin Rädel*, Anna-Janina Bednarek*, Jochen Schmidt†, Christian Willberg*

*Structural Mechanics Department
†Composite Technology Department
Institute for Composite Structures and Adaptive Systems, German Aerospace Center (DLR)
Lilienthalplatz 7, D-38108 Braunschweig, Germany
christian.willberg@dlr.de

Keywords: Peridynamics, Fracture, Mesh dependence.

Summary: Simulations based on the peridynamic theory are a promising approach to understand the processes involved in matrix failure inside fibre reinforced plastics. Before such complex simulations are carried out, the material behavior of bulk resin material as well as the influence of numerical parameters have to be investigated. In the present text, the linear elastic part of the material response is used to examine the convergence behavior of peridynamic simulations. Possibilities to minimize the effect of different discretization schemes are explored by means of a stochastic material distribution in correlation with scatter found in the material tests regarding the elastic material response and failure patterns. This procedure may also be used to investigate the nature of failure initiation and the robustness of the solution.

δ horizon  ρ mass density  θ dilatation  
ε strain tensor  σ Cauchy stress tensor  G shear modulus  
H neighborhood  K bulk modulus  V volume  
Vw weighted volume  ω influence function  t pairwise force density  
x initial bond length  y deformed bond length  e bond extension  
b external forces  u displacements  ü acceleration  
x initial position of reference point  x’ initial position of family point  
y deformed position of family point  T force vector state  Y peridynamic state

1. INTRODUCTION & LITERATURE

1.1 Motivation

Today, the full exploitation of the lightweight potential of fibre reinforced plastics (FRP) is limited due to missing reliability of failure predictions of real structures, especially when taking
into account determining manufacturing conditions. The underlying goal behind this study is
to increase the understanding of failure mechanisms in FRP as shown in Figure 1 and their
numerical simulation.

(a) Crack in a CFRP specimen. Courtesy of DLR.

(b) Matrix failure [1, 2]

Figure 1: Exemplary failure mechanisms in FRP materials

The current state-of-the-art methods used in industry and research for failure predictions
are based on continuum mechanics (CM) and its numerical implementation in the finite ele-
ment method (FEM). The continuum mechanics is well suited for stress analyses of undamaged
structures, but it is unable to proper model damage evolution after initiation. The basic contin-
umum mechanical theory was originally developed around 1822 by Augustin-Louis Cauchy [3].
The assumptions made by Cauchy lead to a mathematical description of continuous media par-
tial differential equations (PDE). With proper restrictions, the PDEs are elliptic in equilibrium
problems. It is to be noted that the underlying boundary value problems are generally well-
posed for typical materials [3]. This made the PDEs solvable even in the pre-computer time. In
reality, all materials are discontinuous and heterogeneous. For several problems, the usual as-
sumption that at macroscopic length scales a material can be well approximated as continuous,
is not valid. Obviously, a fracture in any material fails to satisfy the smoothness requirement.

To overcome this deficit, additional theories such as fracture mechanics are required and
applied. However, certain levels of inconsistencies within the mathematical assumptions be-
tween continuum and fracture mechanics still lead to inaccurate damage prediction. Motivated
by ideas of molecular dynamics, Stewart Silling developed the fundamental peridynamic theory
in the early 2000’s as an alternative theory to state-of-the-art modelling approaches [4]. In this
theory the fundamental PDE of the momentum conservation is replaced by an integral equation.

Peridynamics (PD) presents a promising approach to simulate damage initiation, evolution
and interaction in any material in one holistic approach. It is a non-local theory which takes
long-range forces between material points in a certain neighborhood, the horizon $\delta$, into ac-
count. Constitutive models in peridynamics depend on finite deformation vectors, as opposed
to classical constitutive models which depend on deformation gradients [5]. In contrast to the
FEM based on continuum mechanics, the peridynamic governing equations are based on inte-
gral equations, which are valid everywhere - whether a discontinuity exists in the material or
not. Damage is directly incorporated in the material response.
1.2 Horizon and convergence

A main question in PD simulations is the proper choice of the horizon $\delta$ and convergence based on the chosen discretization $h$, cf. Figure 2. In its basic paper on the meaning of the horizon explains that the value may be viewed as an effective interaction distance for non-local effects [6]. The PD horizon does not have to be constant over the domain. [7] reports that if peridynamics is used to model atomic-scale phenomena, the horizon becomes the cut-off radius of the atomic potential. In the present text an ideally continuous and homogeneous structure is considered, in which the standard local theory applies perfectly until failure. However, we still wish to use peridynamics as a way to model damage and fracture. In this scenario the physics of the interactions between material points do not directly determine the choice of discretization type, size and horizon. [8] found that the two dominant physical mechanisms that lead to size dependency of elastic behavior at the nanoscale are surface energy effects and nonlocal interactions. They estimated the length scales at which the classical model of elasticity breaks down for some real materials. They report that in many materials, the length scale, relevant to forces that determine the bulk properties of materials, far exceeds the interatomic spacing and thus long-range forces contribute to the material behavior. This is particularly true for heterogeneous materials. However, the length scales of interest are still dimensions smaller than the macroscopic behavior investigated in the current context. Thus the question arises: Do discretization, element size and horizon have any influence on the macroscopic failure of the considered structures.

It has been shown in various publications that the classical continuum mechanics is a subset of peridynamics and that for a horizon striving to zero, the peridynamic theory converges to the local solution of continuum mechanics. According to [6] this is since wave dispersion due to the size of the nonlocality is reduced as the horizon decreases. [9] shows the convergence to the local solution for bond-based peridynamics, [10] for an isotropic linear elastic material and [11] show that the state-based, nonlocal peridynamic stress tensor reduces to the classical local Piola-Kirchhoff stress tensor in the limit of a shrinking horizon.

![Figure 2: Types of convergence in the peridynamic theory [3]](image)

The convergence of the discretized implementation is a more complex topic compared to the
The finite element method due to the two independent parameters of element size $dx$ and horizon $\delta$. In [7, 12] the terms $\delta$- and $m$-convergence were introduced, where $m = \frac{\delta}{dx}$ for uniformly discretized grids. [13] also discusses these types of convergence.

Therein it is said that $\delta$-convergence is achieved by fixing $m$ while allowing the horizon $\delta \rightarrow 0$ or increasing $m$ at a slower rate than the decrease in $\delta$. In [14] $\delta$-convergence is described that if the $m$-ratio is kept constant, the solution does not change significantly as the horizon tends to zero. For this type of convergence the numerical peridynamic solution converges to an approximate classical solution. The larger the value of $m$ is, in other words the smaller the grid spacing $dx$ is, the better the approximation becomes. However, this convergence may not occur in the presence of discontinuities. A non-continuous convergence behavior is expected for the variation of the two factors. [12] points out that convergence is dependent on the computation scheme of nodal amount of volume of all points in horizon of single point. Its calculation is not easy to perform for nodes that are not entirely contained inside the horizon. Simple algorithms lead to non-uniform $m$-convergence. The proper choice of horizon has to capture the damage types and the main features of the damage evolution processes and should be performed by means of an absolute length scale, independent of the discretization size [14, 15]. A similar observation was found in [16]. The authors tell that $\delta$ should be at least as large as the crack tip plastic process zone, to adequately capture the crack tip physics. Additionally, some experimental intuition may be required to estimate the size of $\delta$ if local measurements are not available.

Multiple specific horizon values are suggested in different publications over the years. $\delta \approx 3dx$ and thus $m \approx 3$, especially $m \approx 3.015$, is the most common value and is used for example in [7, 17], [18, 19] for micro brittle material, for bond-based composite DCB and ENF specimen [20] and for a finite element representation of peridynamics via truss elements [21]. It has been found that this value of the horizon also works well for fracture predictions [17, 22]. $m = 4$ is applied in [6, 23] for dynamic crack branching problems in isotropic materials and in [24] for flow through porous medium. Even larger values are used in [15] for anisotropic materials with $m = 5$ and $m = 6$ in [14] for linear elastic isotropic material. On the other extreme [25] apply $\delta \approx 1.1dx$ for elastic deformation of thin plate in 2D.

Naturally, the necessity for problem-dependent convergence studies becomes obvious. Examples can be found in [26] for a pitting corrosion problem. [27] focused on the convergence of numerical solutions of static PD problems to the analytical solutions of those problems under grid refinement for uniform grids, while keeping the horizon fixed. The source achieves first-order convergence for smooth solutions. Higher convergence rates can be achieved through higher-order discretizations, quadrature-based finite difference discretizations with piecewise linear basis functions [28] or piecewise linear finite element discretizations [28–30], which leads to a second-order convergence of numerical solutions in PD problems characterized by smooth solutions. These higher-order methods, however, significantly increase the complexity of numerical implementations as well as the computational cost of simulations, especially in higher dimensions. [27] found that achieving convergence is challenging, in particular with respect to the proper choice of horizon. The authors found that, especially in higher dimen-
sions, the horizon cannot be so small as to make computations intractable, but it cannot be too large either as this results in the boundary layer, where displacement boundary conditions are imposed, being the majority of the simulation domain. [31] performs convergence studies for a linear elastic static implementation of non-ordinary state-based PD by means of a zero-energy control term. For $\delta m$- & $\delta$-convergence the authors find, the optimum value increases with increasing mesh size, where the magnitude of the control term increases roughly linearly with the number of degrees of freedom. The $\delta m$-behavior shows first-order convergence independent of the horizon size, whereas the $\delta$-convergence shows approximately half the rate of convergence. Similar findings are reported by [19]. Therein it is stated that choice of the horizon influences heavily the results. The nodal spacing has to shrink faster than the horizon to obtain convergence. Their results suggest that a good nodal spacing can be found for almost all materials for each horizon and vice versa if a small error is acceptable. Unfortunately, no regular pattern was found from which one can determine a simple functional relation between the horizon and the nodal spacing, which makes the choice of a suitable horizon for a given nodal spacing hard. Somewhat contradictory observations are made in [14]. Therein, the error in 2D linear elasticity state-based PD simulations in which the displacement field is linear is not influenced by $\delta$.

A further question is how to discretize numerical implementations in peridynamics. A simple particle-based discretization for the strong form of peridynamic equations was introduced in [17] and is implemented in currently known codes such as EMU and Peridigm. However, [32] point out that the governing equations in peridynamics are continuum models and can be discretized in many ways. In [27] it is mentioned that commonly used meshfree methods in peridynamics suffer from accuracy and convergence issues, due to a rough approximation of the contribution to the internal force density of nodes near the boundary of the neighborhood of a given node. However they are numerically efficient since finite element discretizations of governing equations are based on weak forms, which for peridynamic equations double the number of spatial dimensions that need to be discretized [29].

Approximately uniform element sizes are used in most publications. PD in Peridigm does allow for small gradients in element size if the horizon definition is modified appropriately in the block definition. [22] proposes an adaptive refinement algorithm for the non-local method 1D bond-based peridynamics.

2. PERIDYNAMICS

PD is a non-local theory to describe the physics of materials. Several assumptions made by the classical continuum mechanics theory are weakened or omitted. In continuum mechanics the medium has to be continuous, the internal forces are contact forces and interact in zero distance to each other. The deformation has to be two times differentiable [3]. These assumptions have no physical motivation. In [33] the comparison of the continuum mechanics and the ordinary state-based PD for the linear momentum balance is shown. The main difference from a mathematical point of view is that the PD theory is an integral formulation whereas the continuum mechanical theory is a partial differential equation. Therefore, if the material is discontinuous the continuum mechanics must fail. If the integral domain is zero PD and CM will
be equal, see Equation 1.

Multiple PD formulations exist. The simplest, the bond based (BB) formulation, was presented in 2000 [4]. Therein, materials are limited to a Poisson ratio of $\frac{1}{4}$ for 3D and 2D plane strain problems as well as $\frac{1}{3}$ for 2D plane stress problems [34]. To overcome these restrictions, enhancements of the method have been developed. The so called ordinary (OSB) and non-ordinary state-based (NOSB) formulation of PD are the result.

2.1 Ordinary state-based peridynamics

In the original BB formulations, bond forces only depend on a single pair of material points. The state-based formulation considers bond forces dependent of deformations of all neighboring material points. The state-based PD is able to describe materials loosen the requirements on the Poisson ratio. It must be noted that, within the state-based peridynamic framework, there is no notion of connectivity such as a spring like force between two neighboring material points. There simply is a potential between them. The equation of motion of the OSB-PD is represented as

$$\rho(x) \ddot{u}(x, t) = \int_{\mathcal{H}} \left( \mathbf{T}[x, t]\langle x' - x \rangle - \mathbf{T}[x', t]\langle x - x' \rangle \right) dV + b(x, t) \tag{1}$$

where $\mathcal{H}$ is a spherical neighborhood of radius or horizon $\delta$ centred at $x$ and where $\mathbf{T}$ is the force vector state field. All points $x'$ within the horizon of $x$ are called family of $x$. It maps the force of the bond $\langle x' - x \rangle$ to force densities per volume [35]. The variables $b, \rho, u$ and $\ddot{u}$ are the external forces, the mass density, the displacement and the acceleration.

![Figure 3: Family: initial & deformed configuration with deformation state $Y$ [33]](image)

$\mathbf{T}$ has to ensure the consistency with basic physical principles as the balance of linear momentum. This can be shown for any $\mathbf{T}$. To describe a material, constitutive models are needed. These models map specific deformation vector state fields $Y$ in the force vector state $\mathbf{T}$.

$$\lim_{\mathcal{H}\to 0} \int_{\mathcal{H}} \left( \mathbf{T}[x, t]\langle x' - x \rangle - \mathbf{T}[x', t]\langle x - x' \rangle \right) dV = \text{div}(\sigma) \tag{2}$$
2.2 Material model

It is assumed that the elastic strain energy in a PD solid is equal to the energy of the CM model. In that case, it is supposed that there is a PD strain energy density function $W(\Delta) : V \rightarrow \mathbb{R}$ such, that for some choice of the deformation gradient

$$\mathbf{Y}(\xi) = \mathbf{F} \mathbf{e} = \mathbf{F}(x' - x) \quad \forall \xi \in \mathcal{H}. \quad (3)$$

Then the PD constitutive model corresponds to the classical constitutive model at $\mathbf{F}$ [35, 36]. With the extension scalar state $\epsilon$

$$\epsilon = y - x, \quad y = |\mathbf{Y}|, \quad x = |\mathbf{X}| \quad (4)$$

the pairwise force density for an isotropic elastic PD solid

$$t = \frac{3K\theta}{V_w} \omega x + \frac{15G}{V_w} \omega e^d \quad (5)$$

can be determined utilizing the bulk modulus $K$ and the shear modulus $G$. The variables $\theta$ and the deviatoric part of the extension scalar state $e^d$ are given as

$$\theta = \frac{3}{V_w} \int_{\mathcal{H}} (\omega x) \cdot e dV \quad \text{and} \quad e^d = e - \frac{\theta x}{3}. \quad (6)$$

The value $V_w$ is the weighted volume and $\omega$ is the influence function which can be used to weight the bond stiffness related to the position in $\mathcal{H}$. It is a part of the constitutive response. The complete derivation is given in [35]. The model is similar to the classical one

$$\sigma = K \text{tr}(\epsilon) + 2Ge^d, \quad (7)$$

where $\sigma$ is the mechanical stress, $\text{tr}(\epsilon)$ is the trace of the mechanical strain and $e^d$ is the deviatoric part of the mechanical strain.

2.3 Damage model

One method of introducing failure into PD is through the irreversible breaking of “bonds” by setting the potential between them to zero. Failure is introduced by allowing the removal of this potential when certain physical variables reach a critical level [37].

$$w_c = \frac{4G_0}{\pi \delta^4} \quad (8)$$

The critical micro potential can be determined using the energy release rate $G_0$ in Equation 8. [16, 37] describe an energy-based failure criterion which is valid for state-based analysis by comparison of the critical energy density to the energy density of each state between material points. If the bonds micro potential is greater than this value the bond is deleted. With the history-dependent scalar valued function $\chi(\xi, t)$
\[
\chi(\mathbf{e}(\xi), t) = \begin{cases} 
1 & \text{if } w(\mathbf{e}(\xi)) < w_c \text{ for all } 0 < t' < t \\
0 & \text{otherwise}
\end{cases},
\]  

(9)

the damage model can be included in Equation 5.

\[
t = \chi(\mathbf{e}(\xi), t) \left( \frac{3K\theta}{V_w} \omega x + \frac{15G}{V_w} \omega d \right)
\]  

(10)

Each “bond” has a simple damage law as shown in Figure 4a, whereas the resulting integral material response is illustrated in Figure 4b. It can be seen that the integral behavior corresponds to a standard analytical traction-separation law [3]. The dissipated energy in the material is simply the integral of the bond breakage energies over all the broken bonds in the family [38].

![Comparison of the bond and integral material damage response](image)

Figure 4: Comparison of the bond and integral material damage response

The damage law used in this publication is much simpler. It is assumed that for the one-dimensional cases considered, a critical bond elongation determined in CM can be used as input for the so-called critical stretch criterion, as done by [39] for bond-based peridynamics. In that case Equation 9 is reformulated to

\[
\chi(\mathbf{e}(\xi), t) = \begin{cases} 
1 & \text{if } \left| \frac{y(x', t) - y(x, t)}{|x' - x|} \right| < \epsilon_{\text{crit}} \text{ for all } 0 < t' < t \\
0 & \text{otherwise}
\end{cases}
\]  

(11)

with \( \epsilon_c \) as critical stretch value. [3] point out that this method is derived from BB-PD and that the concept may not apply in state-based material models as used in the current study. However, no other failure model has been implemented in the current numerical framework yet.

3. PROBLEM

In a first step, the behavior of the fibre-embedding epoxy matrix is investigated. Matrix cracking is a dominating mechanism for the failure behavior of the overall FRP material and is most likely to cause other phenomena in the course of damage evolution [2]. Therefore, tensile material tests are performed and evaluated on bulk LY564 epoxy resin tensile specimen,
as shown in Figure 5. The goal is to describe the individual component material properties and failure patterns before application in a more complex structure as shown in Figure 1b.

![Static test rig](image1)

![Effective LY564 stress-strain curve](image2)

![Fracture plane micrograph](image3)

Figure 5: Bulk resin tensile test

3.1 Specimen geometry

The tested structure is a bulk resin test specimen according to DIN EN ISO 527-2 with geometry 1BA. The specimen geometry and its dimensions are shown in Figure 6 and Table 1.

![Bulk tension test dimensions](image4)

Figure 6: Bulk tension test dimensions [DIN EN ISO 527-2, 2012]
Variable & Description | Unit | Standard | Model & Test
---|---|---|---
l_3 | Overall length | mm | ≥ 75 | 75
l_1 | Parallel narrow length | mm | 30.0 ± 0.5 | 30
r | Radius | mm | ≥ 30 | 40.45
l_2 | Distance between wide parallel edges | mm | 58 ± 2 | 58
b_2 | Wide parallel edge width | mm | 10.0 ± 0.5 | 10
b_1 | Narrow parallel edge width | mm | 5.0 ± 0.5 | 5
| Preferred thickness | mm | ≥ 2 | 2
L_0 | Measuring length | mm | 25.0 ± 0.5 | 25
L | Clamp distance | mm | l_2^{+2}_0 | 58

Table 1: Bulk tensile dimensions for test specimen 1BA [DIN EN ISO 527-2, 2012]

The load-displacement curves are measured using strain gauges on one side of the specimen. The resulting unsymmetrical behavior in the area of the strain gauge in combination with the localised change in stiffness in this area leads to failure in the area of the strain gauge.

3.2 Material properties

Low viscosity epoxy resin Araldite LY 564 with Aradur 22962 hardener from Huntsman [40] is used. The material properties can be found in Table 2.

<table>
<thead>
<tr>
<th>Variable &amp; Description</th>
<th>Unit</th>
<th>[40]</th>
<th>Test</th>
<th>Literature</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>ρ</td>
<td>Density</td>
<td>1 · 10^{-9} t mm^{-3}</td>
<td>1.1 – 1.2</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>E</td>
<td>Tensile modulus</td>
<td>N mm^{-2}</td>
<td>2800 – 3300 3190</td>
<td>-</td>
<td>3190</td>
</tr>
<tr>
<td>ν</td>
<td>Poisson ratio</td>
<td>-</td>
<td>-</td>
<td>0.334</td>
<td>-</td>
</tr>
<tr>
<td>ε_u</td>
<td>Failure strain</td>
<td>%</td>
<td>3.5 – 8.0</td>
<td>7.2</td>
<td>-</td>
</tr>
<tr>
<td>G_{IC}</td>
<td>Fracture energy</td>
<td>N mm^{-1}</td>
<td>0.2 – 0.26</td>
<td>-</td>
<td>[41] 0.2</td>
</tr>
</tbody>
</table>

Table 2: Araldite LY 564/Aradur 22962 material properties

In order not to complicate the work in the present study, only linear material response and brittle fracture is considered. The material in the peridynamic simulation with Peridigm is performed using the linear peridynamic solid (LPS) material model.

4. IMPLEMENTATION

4.1 Computational framework

Peridigm is used in the context of the present study [42]. It is an open-source computational state-based PD code developed at Sandia National Laboratories for massively-parallel multi-
physics simulations. Peridigm uses a FE mesh as basis for its discretizations. Hexahedron and tetrahedron elements are transformed into peridynamic collocation points and associated with the respective element volume. Different material properties can be assigned by dividing the model into multiple blocks.

4.2 Stochastic model

To reduce possible dependencies of the solution from the underlying discretization scheme, a stochastic distribution of elastic material properties is proposed to incorporate the statistical nature of damage initiation (Figure 7). Additionally, this gives a possibility to check whether a failure pattern is driven by the chosen discretization or an actual phenomenon. [43] published a similar idea for capturing damage evolution by introducing fluctuations in the critical stretch by means of a Weibull or other distribution. The stochastic distribution of the elastic constants is also motivated by scatter in stress-strain curves and locations of failure of different test specimen and findings in micrographs in the bulk resin specimen (Figure 5). These deviations may be caused by micro-voids, locally varying degree of cure in the epoxy material or slight disparities of the specimen geometries caused by the machining process. Introduction of a stochastic material distribution has the goal to filter and numerical effects in the simulation and to ensure, that the dominating effect causing the physical failure is adequately described in the numerical model. The calculations have to be performed multiple times with different stochastic distributions to assure the dominating effect is adequately triggered.

When Peridigm computes the internal force, it computes a force state at each node in the model and applies that force state to each bond that is attached to the node. For each bond, the resulting force density is applied to the node itself, and negative one times the force density is applied to the node on the other end of the bond. This is consistent with the state-based formulation in Equation 1. The way Peridigm handles material interfaces is basically a direct application of Equation 1. The result at a material interface is an average of the two material models. Thus, a block-based stochastic model is possible by simply assigning materials with
different elastic constants.

As the nature of the distribution of stochastic effects in the real specimen is currently un-known, a rather simple approach is chosen for their modeling. During the creation of the specimen, elements in the damage-prone area are stochastically associated to multiple block definitions. Each block is associated with a material that has a defined deviation from the nominal elastic constants. The number of different block definitions and the maximum deviation from the nominal elastic constants can be chosen randomly. More complex distribution, such as Gaussian or Weibull distribution, may be implemented in the future if the approach seems promising.

5. MODEL

5.1 Discretization

A FE based mesh input is used by Peridigm. As it is expected there are differences in the choice of the horizon and element size for structured and unstructured based meshes, both are considered here. The specimen creation in a versatile parametric model generator allows for a quick change of the underlying discretization scheme and the element size. The base FE models and resulting PD discretizations are shown in Figure 8.

![Figure 8: Discretization schemes and PD representation](image)

The structured mesh is doubly symmetric regarding the specimen $x$-$y$- as well as the $x$-$z$-plane. The unstructured meshes are only symmetric about the $x$-$z$-plane.
Especially in higher dimensions, the horizon cannot be too large either as this results in the boundary layer, where displacement boundary conditions are imposed, being the majority of the simulation domain [27]. Thus, a no-damage zone (red) is introduced in the vicinity of the specimen ends. In this region failure is not modeled to avoid effects of the boundary conditions on the failure behavior. Based on the findings in the experiments this approach is valid.

5.2 Loads and boundary conditions

Both specimen ends are clamped in the test fixture. The tensile experiments are strain-controlled by means of a constant velocity on one of the clamping regions. The homogeneous displacement and inhomogeneous velocity boundary conditions are applied on respective node sets at the specimen ends. As these sets are defined on the base FE mesh, there is a small deviation of the application region in the PD model. This has no effect on the results. Various combinations of displacement boundary conditions were investigated. The influence on the results is negligible.

![Constraint and load introduction domains](image)

Madenci and Oterkus [44] point out, that simply imposing constant boundary condition values on a material regions leads to incorrect behavior of the actual boundary and the domain within a distance of one horizon from the application region. A modified approach to reflect the correct boundary conditions is proposed but not used here as the no-failure-zone in the model is large enough to smooth boundary effects.

6. RESULTS

6.1 Convergence

In a first step extensive convergence studies are carried out. Therefore, only the elastic part of the material behavior is considered. The load-displacement curves of the PD simulations are compared to the solution obtained by the implicit nonlinear solution in the commercial finite element solver Abaqus. Identical meshes are used in both cases. Using the versatile parametric model generator, the identical discretizations are written for Peridigm and Abaqus.
The stiffness convergence is evaluated by means of the load-displacement behavior. Two aspects of convergence are considered. At first it is investigated if the load-displacement curves asymptotically approach a common course. On the other hand, the load-displacement curve from the local FE solution obtained with Abaqus is used as a second convergence criterion for this simple one-dimensional loading condition. It is not expected that the PD solution must necessarily exactly coincide with the FE result. But for this simple test, large deviations should also not occur in the elastic regime of the material response.

6.1.1 Hex mesh

Stiffness Figure 10 shows the respective results for an element edge length $dx$ of 0.4 mm and a structured mesh for different horizons. This element edge length is defined over the thickness of the specimen. Due to the dimensions from Table 1, the in-plane element edge length is 0.395 mm in $x$- and 0.357 mm in $y$-direction. In this study, the horizon is specified by means of an absolute value as proposed by [14, 15] to be able to directly compare the behavior between different element edge length. The non-continuous curves for the PD results are caused by small oscillation in the explicit solution in Peridigm without any damping and a small number of output time steps.

![Figure 10: Force-displacement plot in elastic region for hex-mesh with $dx = 0.4$ mm and various horizons](image)

For the chosen element edge length of $dx = 0.4$ mm, the stiffness reduces with decreasing horizon. This finding does not correspond to the results in [45] where a LPS material model produces a less stiff behavior than expected. The decrease in stiffness is obtained until $m \approx 3$, here $m = 2.95$ for a horizon of $\delta = 1.18$ mm. This matches $m_z = 2.95$, $m_y = 3.31$ and $m_x = 2.99$. The converged PD solution matches the FE solution. If the horizon is decreased below this value, the stiffness rises again compared to the FE solution. This may be caused by
the fact, that not enough neighboring points interact in the horizon of a single point to depict the correct material behavior in all directions, including transversal contraction.

For element edge length above 0.4 mm and thus five PD collocation points over the specimen thickness, a similar behavior exists with the exception that the local FE solution is never reached. For element sizes smaller than 0.4 mm and thus more elements over the specimen thickness a horizon with good agreement can be found for all considered cases.

The results of the study for different element sizes is shown in Figure 11. Since it is impossible to show the load-displacement curves for all combinations in the context of this study, only the relative error of the force at a displacement of 0.1 mm in the load introduction region to the FE solution with an element edge length of 0.2 mm is compared. Results in the upper left corner of the figure are not available as the horizon would be smaller than the element size. The smaller the failure of a combination is, the brighter a point is. A white point corresponds to an error of zero. The minimum combination of each element size and horizon is shown by dashed lines.

It can be seen, that the convergence behavior is not smooth nor continuous. For all considered combinations a value of \( m \approx 3 \) leads to the minimum error compared to the elastic FE response. Also, the error is reduced for finer discretizations. If the mesh is too coarse or the horizon too high, large deviations occur. As expected, the smallest error is achieved for the finest discretization (Figure 12). However, the error for \( dx = 0.4 \text{ mm} \) is sufficiently small and this element size allows a suitable calculation time for the following studies.
Figure 12: Force-displacement plot in elastic region for hex-mesh with $m \approx 3$

**Failure**  According to [46], the choice of the horizon is constrained by a relationship between critical stretch and strain energy release rate. For bond-based peridynamics the respective equation is also given in [3, 17]. [47] claims that a similar equation for state-based model exists. However, the derivation is presumably based on assumptions valid for BB-PD.

\[
\epsilon_{c,BB} = \sqrt{\frac{5G_c}{9K\delta}}
\]

\[
\epsilon_{c,SB*} = \sqrt{\frac{G_c}{\left[3G + \left(\frac{4}{3}\right)^4 \left(K - \frac{5G}{3}\right)\right] \delta}}
\]

For both equations, $\epsilon_c = f(\delta^{-\frac{1}{2}})$. If a critical stretch is chosen for a specific horizon, the critical stretch can be recalculated for any other horizon value by means of this relationship. If the results are compared, for a 1D case, failure should occur at the same displacement. To check this assumption, the load displacement curves for $dx = 0.4 \text{ mm}$ are compared until failure for different horizons (Figure 13).

It can be seen, that Equation 12 does not hold for state-based PD. The specimen fail at totally different displacements, because . A similar pattern as for the stiffness convergence can be seen. This may be caused by the unequal force states of two points in a “bond”. The only proper way to calibrate failure currently is to set the critical stretch to a value where the specimen fails at the same displacement as in the FE simulation or enhance Peridigm by an energy based failure criterion.

In quasi-static loading a symmetrical failure pattern is expected at 4 locations of the specimen. However, a fairly high velocity is chosen to keep calculation times on a manageable level. Due to the combination of explicit time integration without any damping and this velocity it may be possible that inertia effects have an influence on the location of failure. In that case the specimen should fail in the top half, the side with the velocity constraint. The expected
location of failure is not achieved for the converged horizon but for the higher one. However, it is possible that the loading speed is small enough that the damage location is dominated by small numerical effects.

Due to the specimen symmetry failure occurs symmetrically on both sides and evolves in the direction of the specimen mid-plane. The location of failure is comprehensible and lies in the transition to the radius. Mild notch effects due to the change in stiffness and long-range effects of the boundary conditions cause this behavior. A slight kink develops in the crack path, which is most likely to be caused by the discretization pattern and the non-constant PD point volume in the mesh transition domain.

For a 1D stress-state in a BB-PD code [39] proposed that the critical stretch can be taken equal to the maximum principal strain from CM. From a comparison to the Abaqus XFEM solution one can see that this assumption is not valid in the current context. The displacement at failure is even highly unequal between the two discretization types for each converged solution and the same critical stretch.

6.1.2 Tet mesh

Stiffness Similar studies are carried out for a FE base mesh consisting of tetrahedron elements. The results for the same element edge length are not directly comparable between structured and unstructured meshes as the latter consists of a lot more elements for the same edge length. The results of the stiffness convergence behavior are shown in Figure 14.

The stiffness decreases monotonically until the horizon is only slightly larger than one. It can therefore be said, that the unstructured discretization converges to the local FE solution for smaller horizons. The results of all combinations of element size and horizon are shown in
Figure 14: Force-displacement plot in elastic region for tet-mesh with $dx = 0.5 \text{ mm}$ and various horizons

Figure 15 with the same approach as in Figure 11.

Figure 15: Relative force error [%] to FEM solution in elastic region for tet-mesh at displacement 0.1 mm

Convergence is more continuous than for the hex mesh but still far from smooth over different element sizes. The minimum error occurs for a horizon slightly smaller than the element size, here a factor of $m = 1.125$.

**Failure** The results of the force-displacement plots with different horizons and adjusted critical stretch values are shown in Figure 16. The same observation as for the hex mesh can be made. The relationship for the critical stretch from the bond-based PD, Equation 12, does not apply for state-based PD in Peridigm.

The vertical location of failure is at the expected side of the specimen for the converged horizon value. As for the hex-mesh, a higher value of the horizon leads to a more extensive...
failure domain. It has to be noted that in the converged solution, failure occurs at the exact location of transition in the specimen radius. In this localised area the discretization is strongly influenced by the chosen geometrical model. In the present case a subdivision of individual volumes is located there. This seems to influence the failure behavior which is comprehensible for this quasi discrete local model. It seems a quasi continuum nonlocal approach using an unstructured mesh is well suited to capture failure mechanisms.

### 6.1.3 Comparison

For both discretizations at least five elements over the smallest specimen dimension should be used to be able to achieve a convergent solution with negligible errors to the continuum mechanics solution in the elastic regime. The more entropic discretization using a tet mesh and avoiding symmetries in the model leads to a more physical representation of failure. Thus, for more complex studies, the use of an unstructured mesh is proposed.

[11] found that classical elasticity theory is a subset of peridynamics and that PD converges to classical elasticity theory for small horizons. In the present study, and therefore for the numerical implementation of PD, it was found that minimizing the horizon to a bare minimum of \( m = 1 \) only leads to the results of the local finite element method for the unstructured discretization. Structured grids need larger horizon values of \( m \approx 3 \) to assure that enough family members exist so that all directions are adequately covered.

[11] also mentioned that if the only requirement for a peridynamic constitutive model is to reproduce the bulk properties, then horizon is essentially arbitrary. We found that statement to be incomplete as the material behavior is a function of the combination of discretization size and horizon if the behavior is not dominated by small-scale effects.
6.2 Stochastics

The lack of a generally valid failure criterion in state-based PD makes an assessment of the initial idea to use a stochastic material distribution for the assessment of failure initiation difficult. However, stochastics may be used to achieve the same entropy in structured discretization as in unstructured base meshes and to individualize failure locations. The comparison of the original hex model with $dx = 0.4\, \text{mm}$ and horizon $\delta = 1.2\, \text{mm}$ and three models with stochastic material distribution is shown in Figure 17. Ten different blocks are created with a deviation of the 2% of the material bulk and shear modulus.

It can be seen that the overall stiffness and failure behavior does not change significantly. However, the stochastic material distribution makes it possible to spot several possible individual failure locations. One would expect a less slanted crack propagation. This can be achieved by using a finer discretization. The same principal results are valid for tet meshes as shown in Figure 18 with a modulus range of 5% around the nominal value. It can be noted that the location of failure shifts slightly away from the geometric feature bordering the two separate volumes in this region. In one case failure occurs slightly earlier as a result of the stochastic material distribution. Overall, due to the higher mesh entropy, the effect of stochastic material distribution in tet meshes is smaller than in hex meshes.

7. CONCLUSIONS

In the current study, the convergence behavior of peridynamic simulations is investigated using the open-source PD code Peridigm. Multiple base discretization schemes are compared. Different convergence behavior is observed for base hex and tet meshes. While $m \approx 3$ delivers
the best results for hex meshes, \( m \approx 1 \) can be chosen for tet discretizations in case long-range forces have no effect and PD is merely used to improve the simulation of failure compared to CM models.

The use of stochastic material distributions in PD simulations in Peridigm is possible and gives meaningful results. It has proven to be a way to check if the results obtained in PD simulations concerning failure are dominated by numerics and discretization effects or are really the dominating physical effect.

If PD is simply used to model fracture in specimen and conditions not dominated by long-range force effects, the use of tetrahedron base meshes is recommended. The horizon can then be chosen only slightly larger than the element size. Symmetry planes in the model should be avoided. In case a hexahedron mesh is used as an input, a stochastic material distribution is a possibility to increase the model entropy and to get a more consistent prediction of the dominating failure pattern.

The critical stretch damage model must be adjusted to the discretization. The bond-based relationships to the critical strain energy release rate prove unsuited for state-based models. Thus, an energy-based failure criterion will be implemented during the next development steps.

References


[40] *Araldite® LY 564 / Aradur® 22962 data sheet*. Huntsman International LLC. Salt Lake City, Utah, USA, June 2009.


MULTISCALE FIBER KINKING: COMPUTATIONAL MICROMECHANICS AND A MESOSCALE CONTINUUM DAMAGE MECHANICS MODEL

Miguel Herráez*,1,2, Andrew C. Bergan3, Carlos González1,2, Claudio S. Lopes2

*1Department of Materials Science, Polytechnic University of Madrid, E.T.S. de Ingenieros de Caminos, 28040 Madrid, Spain

2IMDEA Materials
C/Eric Kandel 2, 28906 Getafe, Madrid, Spain
miguel.herraez@imdea.org
carlosdaniel.gonzalez@imdea.org
claudiosaul.lopes@imdea.org

3NASA Langley Research Center, Structural Mechanics and Concepts Branch
Hampton, Virginia 23681, U.S.A.

Key words: Fiber kinking, Computational micromechanics, Multiscale

Summary: In this work, the fiber kinking phenomenon, which is known as the failure mechanism that takes place when a fiber reinforced polymer is loaded under longitudinal compression, is studied. A computational micromechanics model is employed to interrogate the assumptions of a recently developed mesoscale continuum damage mechanics (CDM) model for fiber kinking based on the deformation gradient decomposition (DGD) and the LaRC04 failure criteria.

1 INTRODUCTION

There is a need for more accurate mesoscale models for predicting the fiber kinking failure mode in fiber reinforced polymer (FRP) laminates for use in progressive damage analysis (PDA) codes. One factor limiting the accuracy of predictions by many state-of-the-art PDA codes when fiber kinking is active is that most of the physical characteristics of the fiber kinking process are ignored. The conventional continuum damage mechanics (CDM) approach uses the same phenomenological model used commonly for longitudinal tension for longitudinal compression also [1, 2]. The constitutive law typically follows a linear or bilinear softening up to the material point is fully damaged and a stress-free state is reached. Such a model is not reasonable for compression failure modes in FRP laminates since it is not expected that stress will reduce to zero. Instead, it is intuitive that as long as the material remains in contact with itself, some residual stress state will exist as damage evolves.

The fiber kinking theory introduced by Budiansky offers a physically based approach to kink band initiation and propagation [3, 4]. Kinking theory identifies the relevant mechanisms in kink band formation as a combination of an initial fiber misalignment, shear stress-strain behavior that is nonlinear, and large fiber rotation. Fiber kinking theory produces
the characteristic mechanical response shown in Figure 1a where, once the compressive strength, $X_c$, is reached, the stress drops unstably to a plateau crush stress level. The kink band is idealized as shown in Figure 1b with a fiber misalignment angle, $\varphi$, band angle, $\beta$, and band width, $w_{kb}$. If the shear nonlinearity follows a Ramberg-Osgood behavior, $\gamma = (\tau + \alpha \tau^\eta)/G$, Budiansky's theory predicts the strength as,

$$X_c = \frac{G_{12}}{1 + \eta \alpha^{1/\eta} \left( \frac{G_{12} \varphi_0}{\eta - 1} \right)^{\eta}}$$

(1)

where $G_{12}$ is the in-plane shear modulus, $\alpha$ and $\eta$ define the nonlinear shear stress-strain curve, and $\varphi_0$ is the initial fiber misalignment. It is important to notice that this closed-form solution assumes $\beta = 0^\circ$.

Validation of models for fiber kinking is a challenging undertaking due to the instability of the process. Most experimental configurations exhibit unstable failure or interaction of kinking with other damage mechanisms yielding limited data or challenging cases for model validation. In the absence of detailed experimental investigations, computational micromechanics has much utility in providing an alternative basis for evaluating assumptions of models derived at the mesoscale. Recently, high-fidelity three-dimensional (3-D) computational micromechanical models of fiber kinking have been introduced [5, 6] and have shown promising as sources of insights into the fiber kinking process.

In this paper, a computational micromechanical model for fiber kinking is interrogated to evaluate the assumptions of a mesoscale model proposed and further developed by Bergan and Leone [7]. Some of the details of this constitutive model are discussed in Section 2. In the following section, the computational micromechanical model developed by Naya et al. [5] is adapted for use in evaluating the assumptions of the mesoscale model. Finally, in
Section 4 a direct comparison between the two models is presented.

2 MESOSCALE CONSTITUTIVE MODEL

The mesoscale constitutive model represents the fiber kinking phenomenon considering geometric and shear nonlinearities. The model includes the kinematics of the fiber kinking process by tracking fiber misalignment, $\varphi$, throughout loading. The characteristic mechanical response shown in Figure 1a is not prescribed in the model, but is a result of the shear nonlinearity and the fiber rotation. No longitudinal compression fracture energy needs to be specified. The details of the implementation are presented in detail in [7, 8].

The initial misalignment angle, $\varphi_0$, accounts for fiber misalignments and other manufacturing defects that contribute to fiber kinking initiation. Rearranging Eq. (1), the initial fiber misalignment is,

$$
\varphi_0 = \frac{\eta - 1}{G_{12}} \left( \frac{G_{12} - X_C}{X_C \eta \alpha^{1/\eta}} \right)^{\frac{\eta}{\eta - 1}}
$$

Material models that exhibit strain-softening behavior are susceptible to mesh sensitivity when strain localizes. In conventional CDM models, this deficiency is often addressed with Bažant’s crack band theory [9] in which the energy dissipated is scaled by the element size. In the present model, there is no crack surface on which traction goes to zero and therefore the crack band theory is not applicable. Nonetheless, there is an inherent mesh sensitivity since the model includes a strain-softening response leading to strain localization in a band of elements after the strength is reached. The method used herein is analogous to the strain decomposition method by Costa et al. [10] and follows previous work by Bergan and Leone [7]. The model has been implemented in Abaqus/Explicit [11] as a user material subroutine (VUMAT).

![Figure 2. Schematic representation of the model: decomposition into the kink band and the undamaged region](image)
The kink band width, $w_{kb}$, is assumed to be smaller than the element size such that the element can be decomposed into an undamaged material region and a kink band region, as shown in Figure 2. In the kink band region, shear nonlinearity is enabled, whereas in the undamaged material region the shear response is linear. The deformation gradient decomposition (DGD) approach \cite{7, 8} is used to enforce continuity and equilibrium between the undamaged and kink band regions. The DGD approach is enabled (and thus the element is decomposed) when plastic strain becomes non-negligible. However, kink band width, $w_{kb}$, cannot be predicted by the model and it is required as an input. Values reported in the literature range from 50 μm to 200 μm from experimental observations \cite{12-15} and numerical models \cite{5, 6}.

3 COMPUTATIONAL MICROMECHANICS MODEL

In this section, the micromechanical finite element model (FEM) is described including the geometry, discretization, and material properties. Subsequently, a calibration study that was conducted to demonstrate equivalence between the shear behavior of the mesoscale and micromechanical model is described.

A 3-D single-fiber micromechanical model is used to interrogate assumptions of the mesoscale model. The micromechanical FEM is an extension of the single-fiber model described in Naya et al. \cite{5} where it was demonstrated that a single-fiber representative volume element (RVE) produces strength predictions in good agreement with a multi-fiber RVE. The 3-D single-fiber model is used here since the model is a good compromise between computational expense and accuracy. A trade-study between several modeling strategies concluded that a single-fiber and multi-fiber 2-D models were less accurate than 3-D single fiber models. Multi-fiber 3-D models were discarded due to convergence difficulties and computational expense.

3.1 Single-fiber 3-D model for fiber kinking

The model represents a single carbon fiber extruded in the longitudinal $z$-direction along the half wavelength of a sinusoidal curve of length $L$, as shown in Figure 3. The initial misalignment is geometrically introduced according to,

$$y(z) = L \frac{\varphi_0}{\pi} \left( 1 - \cos \left( \pi \frac{z}{L} \right) \right)$$  \hspace{1cm} (3)

$$y'(z) = \varphi_0 \sin \left( \pi \frac{z}{L} \right)$$  \hspace{1cm} (4)

such that the initial misalignment varies along the length of the fiber with the material orientation given by Eq. 4, see Figure 3d.

The fiber diameter is 7.09 μm and the fiber volume fraction is 65%. The model is discretized using 8-node fully integrated isoparametric elements (C3D8). The in-plane mesh size is around 1 μm, while in the longitudinal direction it is 10 μm, and the model length is set to 500 μm. Periodicity of the mechanical fields is guaranteed by the application of periodic boundary conditions (PBC). Gutkin et al. \cite{16} showed that PBC can be applied on
single-fiber models for longitudinal compressive strength prediction, $X_C$, at the expense of inducing $\beta = 0^\circ$. Simulations were conducted in Abaqus/Standard under a dynamic implicit scheme.

![Figure 3](image)

Figure 3. Illustration of the single-fiber 3-D model (a), detail of the mesh (b), exploded cut view of the model (c), and side view with detail of the longitudinal mesh and material orientation (d)

The material system used in this study is AS4/8552 carbon epoxy. Carbon fibers are assumed to behave as linear elastic transversely isotropic solids with the elastic constants shown in Table 1. The matrix behavior is represented using the Lubliner damaged/plasticity model included in Abaqus [11]. This constitutive equation allows the material to behave as quasi-brittle when subjected to dominant tensile stress while it shows elastic-plastic behavior under pressure confinement and compressive loads. The tensile response is, therefore, linear and elastic with modulus and Poisson ratio, $E_m$ and $\nu_m$, until the tensile failure stress, $\sigma_{t0}$, is reached. Beyond this point, a quasi-brittle softening is induced in the material, with $G_t$ being the matrix fracture energy. Under uniaxial compression, the response is linear up to the initial yield limit, $\sigma_{y0}$. Then, stress hardening takes place until the ultimate stress value is reached, $\sigma_{u0}$. The matrix plasticity/damage model parameters used in the simulations are reported in Table 2. Fiber-matrix interface failure is taken into account using a cohesive crack approach. To this end, a cohesive interaction between the fiber and matrix surfaces is defined. The cohesive interaction is governed by a mixed-mode traction separation law where damage onset is controlled by a quadratic stress criterion with normal strength, $N$, and shear strength, $S$. Additionally, isotropic coulomb friction, $\xi$, after cohesive failure is included in the fiber-matrix interaction. The interface parameters used in the simulations are provided in Table 3.
A previous thermal step was applied to introduce residual thermal stresses appearing due to the cooling down process after curing.

### 3.2 In-plane shear response

The mesoscale model utilizes a Ramberg-Osgood shear nonlinearity curve [19]. Ideally, the parameters that define the shear response are obtained from a test that isolates the behavior of a single ply subjected to large shear deformations. However, in the absence of such test data, the ASTM D3518 test of a ±45º laminate subjected to tensile loads is used to define the shear nonlinearity behavior [20]. The ±45º laminate test data smears a wide variety of damage mechanisms into a single stress-strain curve, including large fiber rotations and delamination, which are not desirable to include in the shear nonlinearity characterization. Nonetheless, given the ±45º laminate as the source of material input data for the mesoscale model, calibration of the micromechanical model was performed so that the model produced an equivalent response for an RVE in order to facilitate one-to-one comparison of the two models for the fiber kinking.

An RVE of a ±45º laminate, as shown in Figure 4b, was developed with the parameters and modeling approach described above. The dilatancy angle of the matrix, Ψ, and the temperature drop, ΔT, were adjusted to reproduce the experimental shear curves. The final response achieved from the numerical model, the experimental curve, and the Ramberg-Osgood curve fit are nearly identical as shown in Figure 4a.
4 COMPARISON OF THE MODELS

The predictions from the micromechanical model were compared with a single element analysis using the mesoscale model to understand the role of simplifying assumptions in the latter one. In both analyses, a shortening displacement in the longitudinal direction was prescribed.

The mesoscale finite element is made of one C3D8R element with a uniform edge length of 0.15 mm. The material properties used are provided in Table 4.

<table>
<thead>
<tr>
<th>$E_{11}$ (GPa)</th>
<th>$E_{22}$ (GPa)</th>
<th>$G_{12}$ (GPa)</th>
<th>$\nu_{12}$</th>
<th>$\nu_{23}$</th>
<th>$\alpha$ (MPa$^{1-\eta}$)</th>
<th>$\eta$</th>
<th>$X_C$ (MPa)</th>
<th>$Y_C$ (MPa)</th>
<th>$S_L$ (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>130.6</td>
<td>9.106</td>
<td>4.82</td>
<td>0.3</td>
<td>0.45</td>
<td>2.86 $10^{-11}$</td>
<td>6.49</td>
<td>1400</td>
<td>215</td>
<td>71</td>
</tr>
</tbody>
</table>

Table 4: AS4/8552 material properties for mesoscale model.

The results in terms of stress-strain curves up to the peak load show excellent agreement between the two models for initial misalignments ranging from 0.5° to 4° as shown in Figure 5. The CDM strength prediction follows the expression in Eq. 1, according to the nonlinear shear response described through the Ramberg-Osgood parameters ($G_{12}, \eta, \alpha$).

The stress level (crushing stress) maintained after the peak load plotted in Figure 5a is similar in both models. The sudden load drop observed in the FEM models following the peak load is due to the dynamic nature of the numerical analyses, nevertheless the load stabilizes later and shows an asymptotic response regardless of the initial misalignment. The CDM analyses encountered some convergence difficulties and so not all cases reached the crush stress regime, especially for small initial misalignment angles.
Regarding the kinematics of the fiber kinking process, it is important to differentiate between the maximum, \( \hat{\varphi} \), and the average fiber rotation, \( \overline{\varphi} \), see Figure 6. The maximum fiber rotation takes place in the mid-section of the fiber where \( \hat{\varphi} = \max \varphi(z) \), while the average or global rotation is based upon the deflection of the fiber as, \( \overline{\varphi} = \arctan(u_y/L) \).

The global rotation of the single-fiber model is comparable to the rotation of the CDM model as shown in Figure 7a, while the maximum fiber rotation is higher all along the analysis. It is observed that sudden fiber rotation occurs right after the peak load is reached, then keeps increasing linearly.

From previous experimental studies [13, 21–23], it is reported that once the kink band is generated, it propagates through the specimen at some inclination (kink angle, \( 10^\circ < \beta < 30^\circ \)) with an approximately constant width. However, in the single-fiber model kink band broadening is observed due to the periodic constraints imposed in the model, as shown in
Figure 7b. For this reason, kink band width increases progressively and the representative value for $w_{kb}$ is the one observed right after load drop, which is around $100 \mu m \sim 15 d_f$. The kink band width was one of the input parameters of the mesomechanical model and $w_{kb} = 100 \mu m$ was selected for this analysis.

The effect of fiber-matrix friction coefficient was analyzed by conducting the same simulations with a frictionless interface ($\xi = 0$). The results of this analyses are shown in Figure 8. Although the compressive strength was barely affected, it was observed that there was a drop of approximately 30% in the crushing stress. This drop in stress is explained by the more localized kink band ($w_{kb} = 70 \mu m \sim 80 \mu m$), whose local rotation, $\varphi$, increases up to 30% compared to the reference case with friction. Budiansky et al. [4] related the residual crushing stress, $\sigma_{cr}$, with the in-plane shear strength of the material, $S_L$, and the kink band angle, $\beta$, as:

$$\sigma_{cr} = \frac{2 S_L}{\sin 2\beta} \quad (5)$$

The single-fiber model does not represent $\beta$ due to the periodic boundary conditions. Nevertheless, experimental observations from Vogler et al. [24] showed that $\beta$ is proportional to the fibers rotation, $\varphi$. Thus, larger fiber rotation induces higher kink band angle sustaining lower crushing stress levels.
5 CONCLUSIONS

In this study, a computational micromechanics model is used to assess the assumptions used in the derivation of a continuum damage mechanics model considering fiber kinking failure mechanism. The continuum damage mechanics (CDM) model employs the deformation gradient decomposition (DGD) strategy and is able to capture the main features of fiber kinking such as sudden load drop, kink band formation and rotation, and holds a residual stress level (crushing).

Very good agreement was found between both models not only in terms of compressive strength prediction, but also regarding the kinematics of fiber kinking (rotation and band width) and crushing stress.

REFERENCES


IMPACT RESPONSE OF THICK COMPOSITE STRUCTURES

Niels van Hoorn\textsuperscript{1,\,*}, Christos Kassapoglou\textsuperscript{2}, Wouter van den Brink\textsuperscript{1}

\textsuperscript{1} NLR - Netherlands Aerospace Centre, Marknesse, The Netherlands
\textsuperscript{2} Faculty of Aerospace Engineering, Delft University of Technology, Delft, The Netherlands
\textsuperscript{*} niels.van.hoorn@nlr.nl

Keywords: Impact, thick composites, impact response

Summary: The impact response, in terms of force and displacement histories, is not yet completely understood for thick composite structures. Hence, the goal of this study is to understand how a thick composite plate behaves under impact, assuming no damage, and how this behaviour differs from that of thin composite plates. The elastic response is studied using an analytical impact response model based on a series expansion and a Hertz contact law and compared to numerical results. Small-mass and large-mass 50J impact events are studied. Generally, a good agreement between the analytical and numerical impact response model is found. A sensitivity analysis shows that the response is especially sensitive to impactor mass and sensitive to laminate dimensions. For small-mass impact the thickness is of paramount influence, in contrast to large-mass impact where the area and aspect ratio significantly determine the response. For thick composite structures less energy is converted into bending and more energy is transformed to indentation.

1. INTRODUCTION

Composite materials are known for their high specific strength properties and, unfortunately, for their low damage tolerance. Especially impact damage (e.g. due to tool drops or hail) can be complex and therefore difficult to predict. To compensate for these uncertainties knock-down factors are used which drive the weight and cost of the structure. Accurate damage models might be able to quantify the damage tolerance and result in a better understanding of the damage mechanisms. In turn these models could aid the design and certification process and give a better indication of optimal knock-down factors for impact damage.

Thick composite structures (i.e. 20-50mm or 80-200 layers) are used in highly loaded aerospace structures, such as lugs or landing gear components. For these structures the impact response and damage mechanisms are not completely understood. This paper studies the impact response of a square plate, in terms of force and displacement histories, assuming no damage. Beside the response as a function time, the force versus impactor displacement is a useful way of interpreting the response. This study is part of a an on-going research program on impact damage tolerance of thick composite structures and its goal is to understand the impact response and the differences with impact on thin composite structures before modelling the complex damage mechanisms.
Analytical models that predict the response are available. Some authors that have contributed in this field are Shivakumar [1], Christoforou [2, 3], Olsson [4, 5] and more recently Talagani [6]. The methods they use include energy-balance models to predict the peak force [7], simple spring-mass models that determine the response over time, and more complex models involving series expansions. The spring-mass models can result in efficient closed-form solutions, while the energy-balance models generally require an iterative approach and while more complex models, involving non-linear differential equations, which requires numerical solution techniques. In addition, all these models need a contact formulation that describes the interaction between the impactor and the laminate. The most popular formulation is the Hertz contact law [8], that is frequently used in analytical response models. On the other hand the model of Christoforou and Yigit [9] is very accurate for elasto-plastic contact. Talagani studied several contact formulations in his PhD thesis [6] for different phases, elastic loading, elasto-plastic loading, unloading, and reloading. In this paper one of these analytical response models is selected and modified to study the impact response on a thick composite plate.

The contact formulation is given in Section 2, followed by the analytical and numerical impact response models in Section 3, which are compared in Section 4.1. Subsequently a sensitivity analysis is performed using the analytical impact response model of which the results are given in Section 4.2. A summary and conclusions are given in Section 5.

2. CONTACT FORMULATION

This study uses the Hertz contact law, as given in Equation 1. The Hertz contact law is only accurate in case the indentation ($\delta$) is smaller than the impactor and laminate dimensions [4, 6]. However, in Section 4 it is found that for impact on thick composite plates there is significantly more indentation due to the lack of plate bending. This was also concluded by Talagani [6], who performed an extensive contact study. For the analytical impact response model in Section 3.1 the Hertz contact law is considered sufficient because it is assumed that no damage occurs.

$$F = k_\alpha \delta^{3/2} \quad (1)$$

$$k_\alpha = \frac{4E_z\sqrt{T_i}}{3(1 - \nu_{rz}\nu_{sr})} \quad (2)$$

For the contact stiffness ($k_\alpha$) in Equation 2 it is assumed that the impactor stiffness ($E_i$) is much larger than the laminate stiffness ($E_z$), such that the impactor with radius $R_i$ can be modelled as rigid. In Equation 2 $E_z$ and the Poisson’s ratios $\nu_{rz}$ and $\nu_{sr}$ are derived from the laminate stiffness tensor ($C$). The laminate stiffness tensor is obtained by rotating the ply stiffness tensor to the laminate coordinate system and subsequently averaging all the plies. For a layup that is not transversely isotropic $\nu_{xz} \neq \nu_{yz}$ and therefore $\nu_{rz}$ is determined by averaging the two. In contrast to the contributions of Olsson [4] and Christoforou [3], this is a more accurate description. In their papers it is assumed that $E_z \approx E_{22}$ and that $\nu_{sr} \approx 0$, which are both not the case (see Table 1(a) and 1(b)). Olsson also mentions that Henriksson [10] did a theoretical and experimental study and found that $E_z \approx 1.25E_{22}$ [4]. For the material and layup used in this paper a factor of 1.23 is found, i.e. $E_z = 1.23E_{22}$. 

165
3. IMPACT RESPONSE MODEL

This section presents an analytical and a numerical impact response model. These models predict the impact response of the laminate and impactor in terms of force, displacements and velocity histories. For this study an anti-symmetric balanced transversely isotropic layup is chosen in order to comply with the assumptions in the governing equations. For this layup the B-matrix is zero, as well as the shearing-stretching coupling terms ($A_{16} = A_{26} = 0$) and bending-twisting coupling terms ($D_{16} = D_{26} = 0$).

$$[-45, 45, 0, 90, 45, -45, 90, 0, 90, 45, -45, 90, 0, -45, 45]_n,$$

A fabric material with an approximate 85/15 fibre distribution is used with properties as given in Table 1(a) and the equivalent laminate membrane properties of the above layup are given in Table 1(b). Using the contact formulation of Section 2 $k_\alpha$ is calculated to be 3.48 GPa$\sqrt{m}$. According to the assumption of Olsson and Christoforou (i.e. $E_z \approx E_{22}$ and $\nu_{zr} \approx 0$) the contact stiffness would be 2.67 GPa$\sqrt{m}$, which is a reduction of 23%. This is equivalent to reducing the impactor radius by a factor two of which the effect will be discussed in Section 4.2. The contact stiffness and equivalent laminate membrane properties in Table 1(b) do not depend on the the value of $n$, but the bending properties (e.g. D-matrix) increase with approximately $h^3$. The reference properties used throughout this paper refer to the properties in Table 1 with $n = 5$ (or $h = 20\text{mm}$).

Table 1. The reference properties: (a) the ply properties (85/15 fabric), (b) the equivalent single layer membrane properties, and (c) additional properties related to the laminate or impactor.

<table>
<thead>
<tr>
<th>(a)</th>
<th>(b)</th>
<th>(c)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{11}$</td>
<td>120 GPa</td>
<td>$E_x$</td>
</tr>
<tr>
<td>$E_{22}$</td>
<td>20 GPa</td>
<td>$E_y$</td>
</tr>
<tr>
<td>$E_{33}$</td>
<td>20 GPa</td>
<td>$E_z$</td>
</tr>
<tr>
<td>$G_{12}$</td>
<td>5 GPa</td>
<td>$G_{xy}$</td>
</tr>
<tr>
<td>$G_{13}$</td>
<td>5 GPa</td>
<td>$G_{xz}$</td>
</tr>
<tr>
<td>$G_{23}$</td>
<td>4 GPa</td>
<td>$G_{yz}$</td>
</tr>
<tr>
<td>$\nu_{12}$</td>
<td>0.3 -</td>
<td>$\nu_{xy}$</td>
</tr>
<tr>
<td>$\nu_{13}$</td>
<td>0.3 -</td>
<td>$\nu_{xz}$</td>
</tr>
<tr>
<td>$\nu_{23}$</td>
<td>0.54 -</td>
<td>$\nu_{yz}$</td>
</tr>
<tr>
<td>$t_{\text{ply}}$</td>
<td>0.25 mm</td>
<td>$h$</td>
</tr>
</tbody>
</table>

$$k_\alpha = 3.48 \text{ GPa}\sqrt{m},$$

$$a = 0.2 \text{ m},$$

$$b = 0.2 \text{ m},$$

$$\rho = 1560 \text{ kg/m}^3,$$

$$R_i = 10 \text{ mm},$$

$$E_i = 50 \text{ J},$$

$$m_{i,\text{small}} = 0.04 \text{ kg},$$

$$m_{i,\text{large}} = 4 \text{ kg},$$

$$v_{i,\text{small}} = 50 \text{ m/s},$$

$$v_{i,\text{large}} = 5 \text{ m/s}.$$  

Two types of 50J impact cases are studied, a small-mass 0.04kg (50m/s) impact case (e.g. runway debris) and a large-mass 4kg (5m/s) impact case (e.g. tool drop). In the subsequent sections the reference to small-mass and large-mass impact involves these two cases.
3.1 Analytical Model

The impact response model that is used is based on the model of Christoforou [3]. He assumed that the plate center deflection $w_p$ is described by a series expansion of a modal function. Here $q_{mn}$ is the unknown amplitude and $s_{mn}$ for centrally loaded plates is given in Equation 4.

$$w_p = \sum_{m=1}^{\infty} \sum_{n=1}^{\infty} q_{mn} s_{mn}$$  \hspace{1cm} (3)

$$s_{mn} = \sin \frac{m\pi}{2} \sin \frac{n\pi}{2}$$  \hspace{1cm} (4)

Inserting Equation 3 into the governing equations gives a set of $m \times n + 1$ ordinary differential equations describing the plate and impactor behaviour in the equations below, with the initial conditions $q_{mn}(0) = 0$, $q_{mn}'(0) = 0$, $\delta(0) = 0$, and $\delta'(0) = v_i$. Here force ($F$) is described by Equation 2, $m_p$ is the plate mass and $v_i$ is the impactor velocity. The natural frequencies of a simply supported composite laminate ($\omega_{mn}^2$) are determined using the Classical Laminated Plate Theory, see Equation 7 [6].

$$\frac{d^2 q_{mn}}{dt^2} + \omega_{mn}^2 q_{mn} = \frac{4F}{m_p} s_{mn}$$  \hspace{1cm} (5)

$$\sum_{m=1}^{\infty} \sum_{n=1}^{\infty} \left( \frac{d^2 q_{mn}}{dt^2} s_{mn} \right) + \frac{d^2 \delta}{dt^2} = \frac{F}{m_i}$$  \hspace{1cm} (6)

$$\omega_{mn}^2 = \frac{1}{h\rho} \left[ D_{11} \frac{(m\pi/a)^4}{2(D_{12} + 2D_{66})(m\pi/a)^2 + D_{22}(n\pi/b)^4} \right]$$  \hspace{1cm} (7)

Equation 6 describes the motion of the impactor by Newton’s Second Law $m_i \ddot{w}_i = -F$. Here the relation describing the indentation, i.e. difference in the impactor displacement ($w_i$) and the plate center deflection ($\delta = w_i - w_p$), is inserted together with Equation 3.

The system is reduced to the first order by introducing the set of variables in Equation 8. Substituting these variables and the Hertz contact law in Equations 5 and 6 gives the $2 \times m \times n + 2$ set of first-order ordinary differential equations in Equation 9, with the corresponding initial conditions in Equation 10.

$$q_{mn,1} = q_{mn} \hspace{1cm} q_{mn,1}' = q_{mn,2} \hspace{1cm} q_{mn,1}(0) = 0$$

$$q_{mn,2} = q_{mn}' \hspace{1cm} q_{mn,1}' = \frac{4k_o}{m_p} s_{mn} \delta_1' - \omega_{mn}^2 q_{mn,1} \hspace{1cm} q_{mn,2}(0) = 0$$

$$\delta_1 = \delta \hspace{1cm} \delta_1' = \delta_2 \hspace{1cm} \delta_1(0) = 0$$

$$\delta_2 = \delta' \hspace{1cm} \delta_2' = -\frac{k_o}{m_i} \delta_1' - \sum_{m=1}^{\infty} \sum_{n=1}^{\infty} s_{mn} q_{mn,2} \hspace{1cm} \delta_2(0) = v_i$$  \hspace{1cm} (10)

The system of first order differential equations described above can now be solved using ode45 within Matlab, which is a function based on a variable step Runge-Kutta method. This outputs the indentation ($\delta$) and, using the Hertz contact law, the force as a function of time. The plate deflection ($w_p$) as a function of time is determined by Equation 3 and the impactor displacement by $\delta = w_i - w_p$. 


In Figure 1 the solution is verified by comparing it to the results obtained by Christoforou [3]. The first peak is in good agreement, but there is a discrepancy in the second peak. This is probably due to small differences in the numerical solution procedure and the determination of the plate natural frequencies. Only the first peak is of interest for this study and therefore the solution from this method is considered verified.

The model is validated by comparing the solution with 5J impact experiments on a thin composite plate performed by Lopes [11] and the response model of Talagani [6] in Figure 2. The validation is limited because it considers a low energy impact on a thin laminate, which is outside the scope of this study. The result is, beside a shift in the influence of the plate natural frequency, in agreement with the model of Talagani and in line with the experiments of Lopes.

Convergence is checked by increasing the number of $m, n$ terms in Equation 9. It is observed that for both small-mass and large-mass impact convergence is achieved with 15 terms.

The nature of the response is depends on the involvement of the area affected by flexural waves up to the end of impact ($t_{imp}$). The end of impact, or impact duration, is defined as the moment when the impactor is back at zero displacement. This wave affected area can be determined from the wave front of the first mode, which is described in terms of an ellipse with $r(\theta)$ as in Equation 11 [5, 12]. Here $D_r(0^\circ) = D_{11}$ and $D_r(90^\circ) = D_{22}$ which are the only two values required to define the wave affected area in terms of an ellipse.

$$r(\theta) = \sqrt{\frac{\pi t_{imp}}{h\rho}} \left[ \frac{D_r(0^\circ)}{h\rho} \left( \frac{D_{12} + 2D_{33}}{\sqrt{D_{11}D_{22}}} + 1 \right) \right]^{1/4}$$
3.2 Numerical Model

In parallel with the analytical model a numerical model is developed. ABAQUS/Explicit is chosen because the impact times are relatively short, i.e. in the order of 0.1ms for small-mass impact and 1ms for large-mass impact. During the development of this model different modelling options are considered and weighted in terms of computational time and accuracy.

As a start a layer-by-layer model with continuum solid elements (C3D8R) is created. For this model the ply properties in Table 1(a) and the corresponding orientations are assigned to each ply and all the plies are tied together using a tie constraint. Hard frictionless contact is defined and the mesh is refined near the contact region. Quarter symmetry is applied and it is observed that this does not affect the response in terms of force and displacement. However, the internal stress distribution is affected due to the symmetry because $E_{11} \neq E_{22}$, which will give inaccurate damage predictions.

![Graph](image)

Figure 3. Comparison of the layer-by-layer and single equivalent layer numerical impact response model for (a) small-mass and (b) large-mass impact.

The laminate can also be modelled as a single equivalent layer using the laminate properties given in Table 1(b). The difference between the layer-by-layer and single equivalent layer model is exemplified in Figure 3. For the small-mass impact the results are exactly equal, but for the large-mass impact there is some deviation. The longer impact duration of the large-mass impact event allows the oscillations of the plate to affect the response, resulting in force oscillations. Therefore the natural frequency of the laminate has a significant influence on the result, in contrast to the case of a localised small-mass impact. A change in the properties of the top layer does not affect the contact definition, but representing the laminate as a single equivalent layer with equivalent properties influences the laminate response. In the end the reduction in computational time by more than 60% is considered more important, hence the single equivalent layer model is used.
Several meshing strategies are investigated, for instance a uniform or refined mesh. In the case of the refined mesh the mesh density at the impact location is equal to a uniform mesh, i.e. one element through the thickness, but the mesh density decreases towards the plate edges. There is no noticeable difference in response except for a 92% decrease in computational time for the refined mesh. The convergence of the mesh size is also studied, which is expressed in terms of elements through the thickness \((n_t)\). The results converge for \(n_t = 40\), which is equal to one element for each two plies (0.5mm) and results in an additional 80% decrease in computational time compared to \(n_t = 80\). Finally, the area of the square refined region is increased from \(1\times1\) to \(3\times3\) and \(5\times5\)mm. The results for both small-mass and large-mass impact converged at a refined region of \(3\times3\)mm.

Another popular element for composite laminate simulations is the continuum shell element (SC8R), which has a shell like response but a continuum topology. By comparing these two element types it is concluded that the continuum shell elements have problems with the localised indentation due to a limited through-thickness description compared to the continuum solid elements. Also, a 40% increase in computational time is observed for a small-mass impact case. Due to the above it is concluded that continuum shell elements are not suitable for localised impact cases where large indentations are involved.

4. RESULTS AND OBSERVATIONS

In Section 4.1 the numerical and analytical model are compared. This comparison is a bit arbitrary because in the ideal case both models should be compared to experimental data. However, Figure 2 shows that the analytical model is reasonably close to experimental results. At this point no damage is included in both the analytical and numerical model and therefore it makes sense to compare the analytical and numerical models. In addition to the analytical model based on Christoforou, the small-mass impact model of Olsson [4] and energy-balance model of Esrail [7] are included in the comparison.

4.1 Comparison of Impact Response Models

Figure 4(a) shows the comparison for the small-mass impact event. The analytical impact response model and Olsson’s model yield exactly the same results but show some deviation from the numerical model. This is due to the localised behaviour that is not correctly captured by either the analytical or numerical model. The energy-balance model of Esrail was intended for low-velocity impact and therefore quite far off compared to the other models for a 50m/s impact event. For the large-mass impact event in Figure 4(b) the analytical and numerical impact response model, as well as the energy-balance model of Esrail give similar results. Olsson’s model is based on a single mass spring and therefore not suitable to capture the influence of plate oscillations. For the case in Figure 4(b) the force oscillations between the analytical and numerical impact response model do not match because it is a difficult dynamic phenomenon to capture correctly. For other cases an exact match was observed, but note here that the single equivalent layer description of the numerical model also plays a role, see Figure 3(b).
Figure 4. Comparison of the analytical impact response model discussed in Section 2, Olsson’s small-mass impact response model, Esrail’s energy-balance model and the numerical impact response model given in Section 3 for (a) small-mass and (b) large-mass impact.

4.2 Sensitivity Analysis

The analytical impact response model is now used to perform a sensitivity analysis. In each case a small-mass and large-mass impact event is studied, except for the sensitivity of impactor mass. The key parameters below that determine the response are varied independently com- pared to the reference properties in bold.

- Impactor mass \((m_i)\): 0.04 - 0.25 - 4 kg
- Impactor energy \((E_i)\): 50 - 128 - 200 J (constant \(m_i\))
- Impactor radius \((R_i)\): 5 - 10 - 20 mm
- Laminate thickness \((h)\): 12 - 20 - 40 mm
- Laminate area \((a, b)\): 100 - 200 - 400 mm
- Laminate aspect ratio \((AR)\): 1 - 2 - 4 (constant area)

It is observed that the sensitivity to the impactor mass is high and it can significantly change the impact response, see Figure 5. In addition to the increase in impactor duration, from approximately 0.065ms to 0.35ms and to 0.98ms, the response changes from local (small-mass) to quasi-static (large-mass). In between a typical case of an intermediate-mass impact is shown. The three separate types of impact, i.e. small-mass, intermediate-mass, and large-mass impact can be exemplified by looking at the force and plate deflection versus normalised time, as in Figure 6 for a small-mass and large-mass impact. In contrast to a large-mass impact, the force and plate deflection are out of phase for a small-mass impact. For the intermediate-mass impact the force and plate deflection are still out of phase, but the force history is complex as it can increase and decrease significantly over time simulating multiple impacts, see also Figure 5.
The transition between the response types also involves the wave affected area as discussed in Section 3 and determined by Equation 11. By calculating the impact time $t_{imp}$, it is determined that it takes 0.064ms for the flexural waves to reach the boundaries of the specimen. The impact duration of the small-mass impact case is 0.075ms and it can be seen from Figure 6 that around the time the force is zero at the end of impact the plate oscillations start to occur, which in this case do not affect the response. On the other hand, for the large-mass impact the plate oscillations are visible but the impact duration is too long to significantly affect the response. For an intermediate-mass impact the plate oscillations will significantly affect the response.

The sensitivity to the impactor energy ($E_i$), i.e. the impactor velocity ($v_i$) for a constant impactor mass, is low. However, the force and displacements are increased with approximately the impactor velocity while the impact duration is only slightly decreased. This is expected as the velocity is only an initial condition and can be seen as the amplitude of the system of differential equations. Similar the sensitivity to the impactor radius ($R_i$) has no significant effect on the response shape, but increasing the impactor radius decreases the force and increases the impactor displacement. This effect is similar to scaling the contact stiffness ($k_a$).

The parameters that have a high sensitivity are the laminate dimensions, i.e. the thickness (Figure 7), the area (Figure 8), and the aspect ratio (Figure 9). Increasing the thickness as well as the area doubles the laminate mass and according to Olsson the impactor/plate mass ratio dictates the response [5]. He states that a ratio below 0.23 can be considered small-mass, and above 2.0 large-mass impact. However, it is observed that the sensitivity to these parameters is different. This indicates that the laminate bending stiffness plays a significant role, i.e. increasing the thickness increases the bending stiffness, but increasing the area decreases the bending stiffness. Overall, the area within the force-displacement curve decreases for an increasing thickness. For thicker laminates more energy is absorbed into indentation instead of bending energy during the loading phase. For example, according to the model of Esrail [7], for a thickness of 12mm
about 80% is converted to bending energy compared to 15% for a 40mm thick laminate. Also, for small-mass impact more energy converted back to kinetic energy and therefore the residual velocity is closer to the initial velocity for thicker laminates. For large-mass impact all energy is converted back to kinetic energy resulting in a residual velocity equal to the initial velocity.

In contrast to large-mass impact, increasing the laminate area above $200 \times 200$ mm has no effect on the response for small-mass impact, see Figure 8. The same is observed for the sensitivity to the laminate aspect ratio in Figure 9. It is therefore concluded that in contrast to a localised small-mass impact, large-mass impact is influenced by laminate dimensions and boundary conditions, which is in line with Olsson’s conclusions [5].

Figure 8. Laminate area sensitivity for (a) small-mass and (b) large-mass impact.
5. CONCLUSIONS AND FUTURE WORK

The goal of this study was to understand the impact response of thick composite structures and to identify the differences with impact on thin composite structures before modelling complex damage mechanisms. In an effort to do so an analytical impact response model was developed based on the methodology of Christoforou and Yigit [3] and using the Hertz contact law. A numerical impact response model was also developed for comparison and overall a good agreement between both models was observed. Subsequently a sensitivity analysis on several key parameters was performed, using the analytical impact response model for 50J small-mass and large-mass impact events. This leads to the following conclusions:

A small-mass impact results in a localised response where the force and plate deflection histories are out of phase, in contrast to a quasi-static response due to a large-mass impact. For intermediate-mass impacts a complex response is observed. By increasing the impactor velocity, while keeping the impactor mass constant, the force and displacement are increased by approximately the increase in velocity. For a large impactor radius the contact stiffness is higher and thus a higher force and lower indentation are obtained. For large-mass impact the boundaries and thus laminate dimensions play a significant role. Increasing the laminate thickness increases the bending stiffness of the laminate and, therefore, less energy is absorbed in bending and more in indentation. Similarly as for the thickness, decreasing the laminate area results in less bending, but only for large-mass impact. In addition, the response in terms of plate oscillations can be significantly different. Increasing the aspect ratio, while keeping the laminate area constant, has a similar effect as increasing the laminate area. Again, only for large-mass impact.

The response of thick composite structures can be completely different from the response of thin composite structures. For large-mass impact also the laminate area and aspect ratio that...
define the boundary play a significant role. In general it can be stated that thicker laminates have a higher bending stiffness and therefore less impact energy is converted to bending and more is converted to indentation. In the end, the energy that goes into bending and indentation will result in damage. Thick composite structures generally have a localised impact response which will have a significant effect on the damage that occurs. From literature and previous studies it is known that the damage is mostly internal in the form of matrix cracking and delaminations and maybe a dent at the impact location. As part of an on-going research project, the goal is to predict this damage and identify the differences with impact on thin composite structures.

References


A COMPARATIVE STUDY ON THE MODELING OF MATRIX CRACKS IN FRP PLIES

Ershad Pourbahaaddini Zarandi¹², Philip Simonsson¹, Martin Fagerström¹*, Renaud Gutkin³

¹Chalmers University of Technology
Gothenburg, Sweden
²martin.fagerstrom@chalmers.se
³Norwegian University of Science and Technology (NTNU)
Trondheim, Norway
³Volvo Car Corporation, Durability CAE Body and Trim
Gothenburg, Sweden

Key words: XFEM, smeared crack modeling, matrix cracking.

Summary: In laminated composites, transverse compressive failure of a ply results in a wedge-shaped failure mechanism, formed from matrix cracking, which drives delamination between plies and can initiate local buckling on adjacent plies. In general, continuum damage methods have proven to be efficient in numerical models. However, as a consequence of the inherent homogenization, conventional continuum damage methods such as smeared crack modeling, may have difficulties in adequately capturing these inter-laminar effects due to their inability to replicate the wedge effect. Therefore, in this paper, a discrete method based on the extended finite element method, XFEM, is introduced to explicitly model an inclined crack and to approximate the material response during transverse compressive ply failure. The method is applied in 2D with a predefined inclined crack and a cohesive zone governed by a linear softening traction-separation law. The results of the model are then compared against a similar XFEM approach, with a vertical crack utilizing a transformation in the cohesive zone, and a 3D smeared crack model. Results indicate that the smeared crack approach and the cohesive XFEM approaches successfully predict identical material responses when plies are meshed with one element per ply thickness. By further mesh refinement through the ply thickness, the XFEM approaches show a definitive mesh convergence whereas the smeared crack model is not applicable anymore and predictions become unreliable. Moreover, the proposed cohesive XFEM approach can accurately model the geometrical wedge explicitly, which provides an improved approximation of the wedge effect in comparison to the other two models. In summary, the combined benefits of a discrete XFEM model in terms of mesh objectivity and geometrical accuracy can prove to be significant in future progressive failure analyses involving laminates.

1 INTRODUCTION

The complexity of the failure processes in composite laminates presents a challenging task in predicting global material failure responses. Reliable computational models are required to account for the global effects of individual failure processes that may occur and interact. A typical and significant failure process of interest is that occurring under transverse compressive loading to the fibers. Initiated by inclined matrix cracks, the progressive failure is often linked to initiation and growth of delaminations. Capturing these interacting mechanisms is directly correlated to the accuracy in energy absorption predictions in e.g. crash structures and impact
analyses in applications across automotive, aerospace and naval industries [1, 2].

When modeling composite failure, a relevant scale of observation is the ply level, or the so-called mesoscale, where each ply is modeled separately (but homogeneous) and failure processes such as delamination are distinguishable. To model failure at this scale, continuum damage mechanics methods using the smeared crack approach, such as the model for modeling transverse compressive matrix formation presented by Gutkin and Pinho [3], have shown reasonable peak loads predictions and global material responses. However, Van der Meer and Sluys [4] have shown that continuum models may in some cases not be adequate for the prediction of failure patterns and inter-fiber cracks.

Due to the inherent homogenization, continuum models may have difficulties in explicitly capturing inter-laminar cracking induced by matrix cracks due to their inability to replicate the accurate crack geometries and subsequent deformation pattern. As indicated by Puck and Schürmann [5], a key component in assessing the risk of delamination is to consider the local buckling caused by the wedge effect during transverse compressive failure. Therefore, discrete crack models [6, 7] with the ability to accurately represent the wedge topology may prove to be beneficial.

Considering the number of matrix cracks in a laminate and the possibilities for crack growth analyses, a mesh-independent approach to discrete crack modeling is preferred such as the eXtended Finite Element Method (XFEM), which allows for discontinuities to pass through elements, proposed by Moës et al. [8]. Steenstra et al. in [9] employed such an XFEM based method to capture the traction evolution of an inclined crack and the geometrical wedge effect during pure transverse compression. They did however not explicitly represent the inclined crack geometry. Instead, they employed a strategy where the cohesive law on a vertical crack, perpendicular to the mid-plane of the ply, was transformed onto an inclined fracture plane. This approach is referred to as the transformed XFEM model in this paper.

In the current paper, we present the results of using a fully mesh independent discrete approach based on XFEM, where also the inclination of the crack is explicitly included. The aim is to use this model to explicitly study the material response and prediction of a transverse matrix crack during compressive damage growth, and to compare the obtained results with those obtained both from using the transformed XFEM model [9] as from a smeared crack model [3]. The focus of the paper is indeed on the comparison between the different models, as well as on their capability in producing a wedge topology to account for delamination initiation effects for future studies.

2 THEORETICAL FRAMEWORK

An intact FRP ply in a laminate can with sufficient accuracy be assumed to behave transversally isotropic and linear elastic. However, upon damage initiation the material’s stiffness is assumed to degrade. In this paper, the softening during evolving damage under transverse compression is described by a cohesive zone model with linear softening. In addition, it is assumed that the crack grows throughout the specimen width (W in Figure 1). Thus, one can consider a simplified 2D plane strain model to represent the actual 3D case, see Figure 1, in order to save computational efforts and simplify the analysis. Since the focus is on the middle ply under transverse compression, the model is further simplified by neglecting the effect of the adjacent plies. Hence, the focus is entirely on the formation of the inclined crack.

To evaluate the discrete and continuum models, the framework of the material model proposed by Gutkin and Pinho [3] is adopted for the both cohesive XFEM approaches and the smeared crack model. However, for simplicity and comparison purposes, frictional effects are
not included. It is emphasized that all models apply fixed crack modeling, i.e. the orientation of the crack remains constant during the crack evolution process.

![Figure 1: a) the loaded FRP laminate with the middle ply under transverse compression, b) simplified model in 2D](image)

### 2.1 Proposed XFEM approach with an inclined crack

The XFEM implementation procedure entails the classical standard FE numerical implementations and additional shifted enrichment functions related to discontinuity modeling. In the existence of a crack in the domain, the FE approximation of nodal displacements and test functions are split into continuous and discontinuous degrees of freedom, where the discontinuous degrees of freedom are enriched with a traction contribution. This results in a FE formulation of the problem involving a coupled set of equations expressed in Voigt format, simplified to the case of zero external tractions, as:

\[
\int_{A} B^c \sigma dA = 0
\]

\[
\int_{A^d} B^d \sigma dA + \int_{\Gamma} 2 \bar{N} \mathbf{t} d \Gamma = 0
\]

where \(A\) is the whole domain under consideration, \(A^d\) is the part of the domain in which the discontinuous enrichment is active (in the vicinity of the crack), \(\Gamma\) is the internal surface (with orientation \(\alpha\)) on which the failure process is modeled with a cohesive law governing the degradation of the cohesive traction \(\mathbf{t}\), \(\sigma\) is the Voigt form of the Cauchy stress, \(\bar{N}\) is the Voigt format shape function matrix, \(B^c\) is the Voigt format matrix with shape function derivatives and \(B^d\) is a Voigt format matrix with shape function derivatives multiplied with the enrichment function.

The XFEM model utilizes a level-set function \(\phi\) in the domain \(\Omega\) to determine the position of the crack interface. To be precise, the level-set function is defined as the signed distance function formulated as:

\[
\phi(x) = \pm \min_{x^* \in \Gamma} \|x - x^*\|, \quad \forall x \in \Omega
\]

where \(\|\cdot\|\) is the Euclidean norm and \(x\) is the closest point to \(x^*\) on the crack interface. The enrichment function, to identify and account for crack discontinuity, is then defined as the sign of the level-set function:

\[
\psi(x) = \text{sign}\left(\phi(x)\right) = \begin{cases} 
-1 : \phi(x) < 0 \\
0 : \phi(x) = 0 \\
1 : \phi(x) > 0
\end{cases}
\]
To avoid any numerical issues with so-called blending elements, i.e., elements entailing both standard and enriched nodes but not containing a crack, a shifted sign enrichment is used and the formulation of the discontinuous XFEM approximation of the displacement field is thus given by:

\[ u^h(x) = \sum_{i \epsilon I} N_i(x) u_i + \sum_{i \epsilon I^*} N_i^*(x) \cdot [\psi(x) - \psi(x_i)] a_i \]  

where the first term is the standard finite element approximation across the domain \( I \) and the second term is the added enrichment to the local subdomain \( I^* \) to account for the crack discontinuity. The coefficients \( u_i \) and \( a_i \) are representing nodal unknowns of standard finite elements and enrichments at node \( i \), respectively.

In the finite element implementation of the model, global stresses/forces are transformed from global to local coordinates aligned with a possible fracture plane with an angle \( \alpha \), shown in Figure 2, to compute the resulting tractions used to assess damage initiation in the material. Upon damage initiation, a damage variable \( d \) is introduced to account for material stiffness degradation as the damage, controlled by local shear strains/displacement jumps, develops.

![Figure 2: Global coordinate system and local coordinate system aligned with fracture](image)

The boundary conditions for the model are defined such that the right side of the model is uniformly displaced to the left whilst the displacements on the left side are fixed in the loading direction, see also Figure 3.

### 2.2 XFEM model with transformed cohesive zone law

The transformed XFEM study in [9] uses a similar framework as the proposed XFEM approach. However, the transformed XFEM uses for simplicity a vertical crack (i.e., \( \alpha = 90^\circ \)) to approximate the inclined crack and wedge topology. This is possible by evaluating the cohesive law at an assumed inclined fracture plane, rather than the actual vertical plane, by applying a transformation of the cohesive zone law. Consequently, tractions and displacement jumps are expressed in the inclined \( \{n,t\} \)-frame in the cohesive law, as illustrated in Figure 2, and therefore require an additional transformation between the global and the local fracture plane. Correctly scaled force and material stiffness contributions then allows the modeling of a distinct wedge to form during transverse compression even though the XFEM crack itself is perfectly perpendicular to the loading direction.

### 3 MODEL IMPLEMENTATION

The compressive failure in unidirectional composites is predominantly initiated and driven by the shear along a developing failure plane, as visualized in Figure 2. Therefore, for a 2D case, illustrated in Figure 1 (b), only the shearing component of the traction \( \tau_t \) is used in the damage initiation criterion, which is defined as:
where $S_t$ is the shear strength of the material. The failure initiation index $f$ is then maximized for the range of possible fracture angles $\alpha \epsilon [0, \frac{\pi}{2}]$, and damage initiates once the shear stress reaches the shear strength of the material, when $f = 1$. In the XFEM models, the damage initiation is assumed to take place in the middle of the specimen, while it is not predefined for the smeared crack approach. In addition, the damage evolution after initiation is free to develop in individual elements for the smeared crack models, whereas when failure initiation is detected in the XFEM model a displacement discontinuity surface (line) is introduced through the entire thickness of the considered ply.

After damage initiation, the material degradation is assumed to only affect the tangential stiffness $G$, as no degradation of the material normal stiffness is considered to occur as the cracks are closed and able to carry load during compression. The traction and tangential stiffness at the crack interface is then governed by a developing damage variable $d$. The XFEM models utilize a reformulated cohesive traction-separation law, based on the framework described by Gutkin and Pinho [3], although therein expressed as a stress-strain relation.

For clarity, the main details of the model by Gutkin and Pinho are re-stated here with the traction evolution defined in terms of normal and tangential displacement jumps. Thus the traction on the failure plane is given by:

$$t_{nt} = K \delta_{nt}$$

$$\delta_{nt} = (\delta_n \ delta_t)^T$$

$${t}_{nt} = (\sigma_n \ \tau_t)^T$$

$$K = \begin{bmatrix} k_N & 0 \\ 0 & (1-d)k_T \end{bmatrix}$$

where $\delta_{nt}$ and $t_{nt}$ are the displacement jump and traction vectors respectively. The parameters $k_N$ and $k_T$ are properties that represents the penalty stiffness across the discontinuity in normal and tangential directions respectively. As opposed to the elastic stiffness of the smeared crack model (see [3] for details), $k_N$ and $k_T$ do not have a clear physical interpretation. Instead, they can be considered as numerical penalty parameters used for a convenient implementation of the softening law (including unloading) and should be chosen large enough not to add additional (and unphysical) compliance to the model. However, as can be seen later in this paper, the value of these parameters have a direct influence of the damage evolution rate as function of the displacement jump for the particular choice of cohesive zone model adopted. In fact, this is true even if their values are chosen large enough not to influence the resulting force-displacement relation.

To relate the damage evolution defined for the smeared crack approach in [3] to the displacement jumps, the strains are substituted by displacement jumps, the damage variable is expressed as:

$$d = 1 - \frac{\delta_i}{\delta_t} \frac{(\delta_f - \delta_t)}{(\delta_f - \delta_i)}$$

where $\delta_i$ and $\delta_f$ represents the corresponding discontinuous jump required for damage initiation and complete failure, respectively. Complete failure occurs when $d = 1$, and the
equivalent displacement jump $\delta_f$ for the sliding failure mode (mode II), is defined as:

$$\delta_f = 2 \frac{(G_{IIc})}{S_t}$$

where $G_{IIc}$ is the fracture toughness, and $S_t$ is once again the shear strength corresponding to the damage initiation.

### 3.1 Abaqus set-up for the smeared crack model

The case shown in Figure 1(b) is modeled in Abaqus/Explicit to predict the material response using a smeared crack approach. Three mesh sizes with 1, 3, and 6 elements per ply thickness (0.2 mm), using linear cubic continuum elements C3D8R with reduced integration scheme and without nonlinear geometry effects, are considered in the mesh convergence study. The part contains only one row of element in the fiber direction (direction 1 in Figure 1). A plane strain condition is applied and the driving displacement condition on the right edge is enforced at such a low speed that the simulation can be considered to be quasi-static, and thus, comparable with the results from the XFEM approach. In addition to the aforementioned boundary conditions on the left and right edges, all degrees of freedom of a node at a corner on the left face are fixed to prevent rigid body motion. In addition, the out of plane displacement (the displacement in the fiber direction) is prevented in order to simulate the material in the middle of the specimen width.

A user defined subroutine (a so-called VUMAT routine) based on the framework of Gutkin and Pinho [3], is employed to account for the damage initiation and evolution.

The material properties used in all three models are taken from [3] and listed in Table 1.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal elastic modulus $E_{11}$ [GPa]</td>
<td>128</td>
</tr>
<tr>
<td>Transverse elastic modulus $E_{22}$ [GPa]</td>
<td>7.9</td>
</tr>
<tr>
<td>Shear modulus $G_{12}$ [GPa]</td>
<td>3</td>
</tr>
<tr>
<td>Poisson’s ratios $\nu_{12}$</td>
<td>0.3</td>
</tr>
<tr>
<td>Poisson’s ratios $\nu_{23}$</td>
<td>0.4</td>
</tr>
<tr>
<td>Transverse shear fracture toughness of the material $G_{IIc}$ [KJ/m²]</td>
<td>2</td>
</tr>
<tr>
<td>Shear strengths $S_t$ [MPa]</td>
<td>60</td>
</tr>
<tr>
<td>Density $[Kg/mm^3]$</td>
<td>1.6 e-6</td>
</tr>
</tbody>
</table>

Table 1: Material properties of T700/MTM57
4 NUMERICAL RESULTS

4.1 Comparison of the XFEM approaches

It is clearly seen from Figure 4, where the resulting reaction force at the edge is plotted against the displacement of the rightmost edge, that the XFEM models with inclined and transformed crack can predict the material response identically regardless of the mesh discretization. Furthermore, the proposed XFEM model with an inclined crack predicts the wedge topology explicitly, in contrast with the transformed XFEM, which can generate a wedge however with a vertical crack. Therefore, both methods can produce two distinct yet different geometrical wedges regardless of mesh discretization, see Figure 5.

![Force-displacement curves for the XFEM approaches with inclined and transformed crack](image1)

Figure 4: Force-displacement curves for the XFEM approaches with inclined and transformed crack

![Mesh deformation of the XFEM model with a) transformed, and b) inclined crack](image2)

Figure 5: Mesh deformation of the XFEM model with a) transformed, and b) inclined crack (the damaged elements are colored in red)

4.2 Comparison of the proposed XFEM approach against the smeared crack approach

To compare the results from XFEM with the smeared crack approach, the displacements from the XFEM approaches are normalized over the length of the damaged area across the crack in the loading direction to give strains comparable to those from the smeared crack model, which are computed over the entire domain length. Furthermore, the reaction forces are divided by the specimens’ cross-section areas to provide the normal stresses. The comparison of stress-strain curves resulted from the XFEM approach with an inclined crack
and the smeared crack approach reveals that the latter predicts the material response correctly when mesh consists of one element per ply thickness. The prediction of the model for finer meshes starts to deviate from the XFEM predictions soon after damage initiation, and a significant amount of artificial hourglass energy is detected which has great impact on the results reliability, see Figure 6. We choose herein to define that results are considered as unreliable if the ratio between artificial energy (AE) and the strain energy (SE) exceeds 5%. The kinetic energy is monitored during the simulation so that it does not exceed 5-10% of the internal energy and the simulations remain quasi static.

The predicted deformations from the transverse compression are illustrated in Figure 7 for the proposed XFEM and smeared crack approach. Note that the damage and deformation of the proposed XFEM approach is more localized in comparison to the smeared crack approach. In addition, the mesh refinements XFEM do not affect the deformations in Figure 7, nor the predicted force-displacement response in Figure 6.

For clarity; the (a) case comparisons between XFEM and the smeared crack approach, consisting of one element per ply thickness, as seen in Figure 7, are visually different. This is due to the modeling definitions used in XFEM for boundary conditions. Nevertheless, both approaches have a single element through the ply thickness, and the results presented for XFEM (a) case are for the middle element. This point is described further in discussion.

As an additional note, the mesh discretization for XFEM is adapted to the smeared crack mesh refinements for comparison purposes, as the XFEM model does not require square elements in order to function.
Note that the damage in the XFEM approach is growing much faster than the smeared crack approach, and that well above 80% of the material is damaged soon after damage initiation, see Figure 8. As can be seen however from Figure 9 below, where two lower values of the stiffness $k_T$ have been used ($k_T = 5E_{22}$ and $k_T = 10E_{22}$ respectively), the rapid damage evolution is directly linked to this cohesive stiffness. Upon lowering the penalty stiffness, the damage evolves slower and more similar to the smeared crack case. However, this occurs at the expense of an altered overall force-displacement results due to the introduction of additional compliance into the model as discussed above in Section 3.

Figure 8: Stress-strain and damage evolution curves for the XFEM (with $k_T = 20E_{22}$ i.e. the lowest $k_T$ value resulting in a converged material response) and the smeared crack approach for the mesh with 1 element per ply thickness

Figure 9: Variation of the cohesive stiffness parameter for XFEM with $k_T = 5E_{22}$ (left) and $k_T = 10E_{22}$ (right)

5 DISCUSSION

One key aspect of matrix cracks formed under transverse compression, is the forming of a wedge that can initiate and drive delamination growth. Figure 5 shows that both XFEM approaches are able to produce separate distinct wedges. It is worth noting however, that the shape and location of the wedge tip from the damaged elements are significantly different, where the proposed XFEM approach explicitly predicts an inclined matrix crack topology in contrast to a vertical crack topology predicted by the transformed XFEM approach. Consequently, the differences in local buckling induced by the wedge tips may affect crack formations, which may be important in compressive failure studies in laminates. The precise
locations of local buckling may have cascading effects in the global material response, as they initiate and drive delamination. In addition, as the global crack results in an inclined topology, the proposed XFEM model can explicitly model the crack formation through the ply thickness similar to actual matrix cracking. However, this may prove to be difficult with the transformed XFEM model, where the elements with vertical cracks would then be required to be positioned locally along the inclined global crack topology (i.e. not stacking above each other), which is beyond the model’s scope. This also implies that modeling progressive through-thickness growth of a matrix crack under compressive loading would be more or less impossible using this transformed XFEM model.

From Figure 7, transverse deformation of the proposed XFEM approach successfully generates a detailed geometrical representation of a distinct wedge regardless of mesh discretization. The smeared crack approach however, shows difficulties in capturing shearing deformations as the crack is smeared over the entire damaged elements’ volume. Yet, more elements through ply thickness does indicate that the smeared approach could geometrically represent a deformation resembling a wedge as the crack is resolved further. However, resolving the crack also means that the assumptions of continuum damage mechanics are not valid anymore and causes the predictions of the model to be unreliable.

In general, the smeared crack approach is considered mesh objective, as it is mesh independent for in-plane mesh refinements. However, for comparison purposes with XFEM, the mesh is refined through the ply thickness and the limits of the theory behind the smeared crack approach are tested. As seen in Figure 6, the smeared crack approach is able to predict the material response without hourglassing issues when the mesh discretization is one C3D8R element per ply. However, when the crack is further resolved, the material response prediction diverges and artificial energy increases due to increased hourglassing, which causes the model prediction to be unreliable soon after the peak load. On the other hand, the XFEM results for different mesh discretization are identical, which indicates the XFEM is mesh objective and able to resolve the crack.

Both the proposed XFEM and the smeared crack approaches exhibit near identical predictions in the bilinear material response even though the damage evolutions are drastically different, as seen in Figure 8. The differences in damage evolution can be attributed to the differing kinematics of the two approaches, and to the adoption of penalty stiffness parameters for the cohesive zone model adopted in the XFEM case. For the smeared crack approach, the damage evolution is driven by the shearing strain along the crack surface and grows more gradually due to kinematical constraints by the bulk behavior of the finite element model. In contrast, XFEM explicitly describes kinematics at the crack interface through a cohesive zone model, where the damage evolution is instead driven by the displacement jump in the shearing crack direction, as illustrated in Equation (8). The shearing displacement jump at the crack then causes the damage to localize and grow more rapidly. In addition, since the adopted cohesive zone formulation is such that the damage variable is used to “scale down” a trial shear stress given by \( \tau^\text{trial} = k_T \delta_t \) to the “correct” traction value on the softening branch (according to \( \tau_t = (1 - d)\tau^\text{trial} \)) of the cohesive zone for a given displacement jump, \( d \) inherently needs to evolve faster if \( k_T \) is larger. This in order to generate the same resulting traction-separation law with the same amount of energy dissipation. To overcome this dependence on the penalty parameter(s), a cohesive zone formulation analogous to standard damage elasto-plasticity could be adopted, as done e.g. in [10]. With such a formulation, the damage evolution would be independent of the value of the elastic penalty parameter(s). Alternatively, a purely softening cohesive zone law, without the initial elastic response (and without elastic penalty parameters), could be adopted. Obviously, neither in this case would the damage evolution
depend on a numerical penalty parameter since such parameters would simply not exist. However, both these suggested alternative formulation would require a completely different implementation and has been considered as out-of-scope for the current paper.

In the presented frictionless case, there are no implications towards differing damage evolutions influencing the model predictions. However, additional internal studies for a single element per ply including frictional contributions, an implementation of the complete framework of the material model proposed by Gutkin and Pinho [3], have indicated that there is a significant deviation in model predictions of the peak load due to the difference in kinematics and damage evolution. In the full version of the model by Gutkin and Pinho [3], frictional forces acting on the crack surface is assumed to be directly proportional to the damage variable, illustrated in Figure 8, which means that approximately 80% of the frictional forces are applied almost instantly after damage initiation in the XFEM approach. Thus, the appropriate modeling of damage evolution and its interplay with frictional effects still remains to be addressed in future work, especially when a discrete crack modeling approach is adopted.

The normalization of the displacements from the XFEM approaches over the length of the damaged area in loading direction provides strains comparable to those from the smeared crack model. This assumption is valid for the XFEM approaches as the strains in the outer (intact) elements in loading direction are negligible. Thus it can be assumed that the same forces and displacements applied at the boundaries are applied on the boundaries of the damaged area. For the smeared crack model however, this is not true due to the bulk behavior of the finite element model. This is even more pronounced if frictional effects are considered, where damage then spreads into a larger portion of the model’s volume. Thus, the bulk behavior of the entire domain needs to be considered, and strain is calculated accordingly.

6 CONCLUSION

Modeling of matrix cracks in FRP laminates under transverse compression has been conducted by three approaches; two based on XFEM (discrete crack modeling) and a smeared crack approach (a continuum damage model). The material model developed by Gutkin and Pinho [3] is implemented as the common framework in the comparison between the approaches and presented here without frictional effects.

The mesh refinement study through the thickness of the ply indicates that the approaches based on XFEM are able to resolve the crack effectively and provide unique results regardless of mesh discretization proving to be mesh independent. The smeared crack approach successfully predicts the material response with one element per ply, but shows different prediction instabilities as the crack is further resolved through mesh refinement. However, considering the fact that the investigation of failures in composite plies is normally conducted on the meso-scale, i.e. ply thickness scale, both approaches with a mesh including only one element per ply are able to produce adequate results.

The results with one element through the ply thickness indicate that, while the damage evolution is significantly different in the proposed XFEM and the smeared crack approach due to differing kinematics, they both predict a nearly identical material linear stiffness softening and energy dissipation during the damage process when frictional effects are excluded.

The mesh deformations indicate that the smeared crack approach requires more elements through the ply thickness to represent a geometry resembling a wedge. However, due to high hourglassing effect, the material response predictions for finer meshes are not reliable. The transformed XFEM approach is not able to properly address this issue either. The proposed XFEM approach however, successfully predicts the geometrical representation of matrix
cracks formed under transverse compression that results in a distinct explicit wedge, regardless of mesh discretization, which is of interest for further studies on FRP laminate failures concerning delamination, crack propagation and crack migration.

7 ACKNOWLEDGEMENT

The authors acknowledge partial financial support from the Swedish strategic innovation programme LIGHTer provided by VINNOVA (grant no. 2014-06041) and from the Hugo Heyman research fund at Chalmers University of Technology.

REFERENCES


MONITORING OF FATIGUE DAMAGES IN ADHESIVELY BONDED JOINTS

Andreas Preisler*, Zamaan Sadeghi*, Kai-Uwe Schröder*

*RWTH Aachen University, Institute of Structural Mechanics and Lightweight Design
Wüllnerstraße 7, 52062 Aachen, Germany
andreas.preisler@sla.rwth-aachen.de

Keywords: Structural Health Monitoring, Structural Damage Indicator, Single Lap Joint, Fatigue, Finite Element Method

Summary: Adhesive joints provide an appropriate joining method for composite materials. Therefore, they are attracting increasing interest especially in the transportation sector (aerospace and automotive). However, adhesive joints still possess several uncertainties in particular concerning fatigue and ageing. This contribution presents a Structural Health Monitoring approach based on structural damage indicators. Deepened structural analyses provide the optimal damage indicator for single lap joints - a zero strain point on each adherent back face. In case of a damage, the former zero strain point is exposed to strain. Thus, measuring strain at this certain spot provides a distinct damage indicator. The applicability of the zero strain point as a structural damage indicator for fatigue damages is verified within numerical fatigue analyses.

1. INTRODUCTION

In general, adhesive joints provide a very efficient joining method for composite materials. In case of adhesive joints, there are no holes, which have to be drilled and therefore fibres do not have to be cut. Load transfer is distributed within the overlapping length. Thus, stress concentration does not appear within the adherents. Due to those reasons, adhesive joints are considered to be an appropriate joining method for composite materials. Various adhesive joints are designed as single lap joints (SLJ), e.g. the joint of two aircraft fuselage sections. Despite the increasing interest in adhesive joints, analytical methods are still used within industry. However, analytical models usually predict failure load well for a certain material of a certain thickness. Especially if thickness of the adhesive joint is increased, the failure load is overestimated [1, 2]. In addition, manufacturing defects (voids, kissing bonds) and insufficient knowledge concerning fatigue and ageing increase the difference between the predicted adhesive behaviour and the real one [3].

In case of a damage, the fail safe design philosophy [4, 5] considers load redistribution due to failure. If one part of the structures fails, the intact structure must be able to withstand all loads. This might be achieved by joining an adhesive with a redundant rivet joint. A damage tolerant design [4, 5] does not consider an alternative load path. However, a damage is tolerated
up to a predefined critical damage size. Below the critical damage size, a damage does not lead
to failure of the structure. In this case, it has to be ensured, that the damage is detected before
it becomes critical. Thus, the damage tolerant design makes a compromise between structure
weight and inspection and maintenance costs.

Structural Health Monitoring (SHM) is the automatic and on-line observation of structural
integrity during service [6]. The implementation of SHM within damage tolerant design is a
logical next step. SHM provides essential knowledge about structural behaviour and the damage
state. A rather simple SHM detects abnormal structural behaviour and possibly a rough location
of the potential source of that behaviour. In this case, a visual inspection for damage assessment
is triggered. A more comprehensive SHM system assesses a detected damage without additional
visual inspection. If a damage is detected before it becomes critical, damage propagation can
be monitored, while maintenance is scheduled.

Various SHM approaches have been proposed to monitor damage within adhesive joints.
Malinowski et al. [7] analysed electro-mechanical-impedance (EMI) of piezoelectric transduc-
ers bonded to the adherents, which resonate through thickness. Their studies indicated, that
the conductance and the frequency shift of the resonant mode might be used as damage indica-
tor. Dugnani et al. [8] manufactured SLJs with embedded piezoelectric transducers within
the bond-line. The cross-correlation of EMI signals was used to track relative changes in fre-
quency and magnitude of the peak conductance. A damage was detected at 60% of fatigue life
(which is specified as 47500 cycles). However, the authors stated that the optimal position of the
transducer within the bond-line is ambiguous. A transducer close to the edge usually detects a
damage earlier. However, asymmetric damage cases are not properly covered with a transducer
close to one edge.

Besides EMI, other approaches based on the mechanical behaviour of adhesive joints are
considered as well. Poveromo [9] described a method based on quantitative percussion diag-
nostics for flat composite panels. This approach is implemented by impacting the structure with
a steel rood containing a force sensor and is based on testing procedures for dental implants. In
case of a weak bond, impact force is reduced due to the reduced stiffness. Galea [10] monitors
the strain ratio of a repair patch and the base structure. If a damage is present within the joint
of the repair patch, load transfer is shifted and therefore, an alteration within the strain ratio is
present.

SHM methods based on the mechanical behaviour have been investigated for decades, util-
ising different damage indicators. However, in many cases sensitivity is questionable or the data
only indicates the presence of a damage without enough information for damage assessment.
This contribution introduces an approach based on sophisticated structural analyses in order to
optimise sensitivity of a damage indicator. The approach is verified within fatigue studies.

2. OPTIMISATION OF STRUCTURAL DAMAGE INDICATORS FOR SINGLE LAP
JOINTS

Analysing the structural behaviour is very beneficial prior to setting up an SHM system.
Structural analyses give an insight into highly stressed regions, so called hot spots, which are
prone to damages. Potential damage mechanisms can be derived based on the stress state. If the expected damages are implemented within the models, the models are capable of characterise the influence of the damage. Comparing both, the undamaged and the damaged model, the change in structural behaviour related to the damage may be analysed. The major change due to damage usually provides very efficient damage indicators, where a sharp and clear damage signal is achieved. Due to their relation to the structural behaviour those damage indicators are hereafter referred to as structural damage indicators (SDI) [6].

SDIs are sensitive to all damages, which have an influence on the load carrying behaviour. In case of a crack, compressive loads closes the crack and thus the crack may transfer compressive loads via contact forces. However, if a tensile force is applied, the crack is opened and stresses are redistributed. Consequently, the structural behaviour differs from the expected behaviour of an intact structure. The severity of that alteration and thus the measured value of the SDI are dependent on the damage size. Therefore, an SDI may provide an intrinsic damage assessment, e.g. using predefined thresholds.

Usually, a damage indicator

\[ SDI_{\text{abs}} = X(x, t) - X_0(x), \]  

is defined as the absolute deviation of the measured quantity \( X \) at the position \( x \) and time \( t \) [6]. \( X_0(x) \) describes the expected value of the intact structure (reference value, baseline), and \( X(x, t) \) the actual measurement at time \( t \). However, the absolute deviation does not give any information about sensitivity of a damage indicator. More insight is gained, if a relative damage indicator

\[ SDI_{\text{rel}} = \frac{X(x, t) - X_0(x)}{X_0(x)} \]

is considered [6]. This relative damage indicator describes the sensitivity of damage indicator. Eq. (2) indicates that a reference value \( X_0(x) \approx 0 \) is very beneficial. In this case, a relatively small change of the absolute value leads to a strong signal. In case of a zero signal, no damage is present. If a non-zero signal is measured, a damage is detected. The amount of deviation usually refers to the damage size.

In the following, this contribution transfers the approach of SDIs, which originally was developed for beam structures [6], on SLJ. SLJ are widespread, since most adhesive joints are designed as a simple overlapping joint (e.g. the joint of two aircraft fuselage sections). Considering a SLJ, a characteristic stress distribution can be derived (see Fig. 1 (left)). Peel stress peaks arise at the edges of the adhesive due to bending deformation. Thus, the edges of the adhesive are prone to damage. Alongside the characteristic stress distribution within the bond-line, the resulting normal strain distribution at the back face can be derived (see Fig. 1 (right)). The consideration of the back face strain enables a rather simple strain measurement without introducing a disturbance within the bond-line.

Zhang et al. [12] utilise strain at the end of the overlapping area to detect damages. In case of a damage, bending is increased within this area. Due to negative bending strain, the
measured strain at this position is decreased. Graner Solana et al. [13] improve this approach by shifting the monitored point towards the region with maximum gradient for the strain curve $\varepsilon_{xx}(x)$. Therefore, Graner Solana et al. monitor strain at the position with the maximum absolute difference as indicated by Eq. (1). Previous studies [11] revealed the high potential of monitoring strain at the zero strain point, which has the highest maximum relative difference as indicated by Eq. (2). At this coordinate, positive strain due to the applied load is nullified by the negative bending strain. This point is load independent and only affected by damages. In case of a damage, load transfer is shifted and hence, the zero strain point is shifted as well.

3. NUMERICAL MODEL

The proposed SDI for a SLJ is investigated within a Finite Element (FE) model under fatigue loading. The SLJ used in the present work is a joint with aluminium adherent (2024 T3) with FM 73M OST toughened epoxy adhesive which was tested under static and fatigue loading in a previous study by Khoramishad et al. [14]. In the following, the FE model developed for the SLJ under static and fatigue loading is explained.

3.1 Static model

Based on the geometry of the SLJ shown in Fig. 2, a 2D FE model is generated in ABAQUS. 4-node bilinear plane stress quadrilateral elements with reduced integration (CPS4R) are used for the aluminium adherents. The aluminium is modelled by a linear elastic, linear plastic material behaviour with hardening. In this way, non-linear material behaviour is described by linear approximation between yield strength and ultimate strength. The material data is summarised in Tab. 1.

4-node two dimensional cohesive elements (COH2D4) are used to model the fracture be-

Figure 2. Geometry of the SLJ (all dimensions are given in mm) [14]
haviour of the adhesive joint under static loading. A bilinear traction-separation law is considered for degradation of the cohesive elements. Damage initiation is predicted by a maximum nominal stress criterion

$$\max \left\{ \frac{t_n}{t_n^0}, \frac{t_s}{t_s^0} \right\}$$

in which $t_n$ is the traction in pure normal direction and $t_s$ is the traction in shear directions. The parameters $t_n^0$ and $t_s^0$ are the corresponding maximum traction values. The Macaulay bracket $\langle \rangle$ is added to ensure that pure compression state has no influence on damage initiation. As for the damage propagation, Benzeggagh-Kenane law [16]

$$G_n^C + (G_s^C - G_n^C) \left( \frac{G_s}{G_n + G_s} \right)^2 = G_n^C + G_s^C$$

is used. Where $G_n$ is the strain energy due to normal traction and $G_s$ is the strain energy due to shear traction. The parameters $G_n^C$ and $G_s^C$ are the corresponding fracture energies. The material properties used for the cohesive elements, representing the adhesive layer, are listed in Tab. 1.

A refined mesh of $0.2 \cdot 0.2 \text{mm}^2$ is considered within the overlap area. The mesh size is transferred to a coarser mesh for the adherent material beyond the overlap. Clamped boundary conditions are applied to a reference node at both ends of the SLJ. The reference node transfers the reaction forces to all nodes at the edge by means of a kinematic coupling. The mesh and the boundary conditions defined are shown in Fig. 3.

### 3.2 Fatigue model

For fatigue lifetime prediction of SLJ, a strain based damage model, which was proposed in literature by Khoramishad et al. [14]. The proposed damage model is expressed as

$$\Delta D \Delta N = \begin{cases} \alpha (\varepsilon_{\text{max}} - \varepsilon_{\text{th}})^\beta, & \text{for } \varepsilon_{\text{max}} > \varepsilon_{\text{th}} \\ 0, & \text{for } \varepsilon_{\text{max}} \leq \varepsilon_{\text{th}} \end{cases}$$

with

<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$E$ - Young’s modulus</td>
<td>$E$ - Young’s modulus</td>
</tr>
<tr>
<td>74500 MPa</td>
<td>2000 MPa</td>
</tr>
<tr>
<td>$\nu$ - Poisson’s ratio</td>
<td>$G$ - Shear modulus</td>
</tr>
<tr>
<td>0.33</td>
<td>724 MPa</td>
</tr>
<tr>
<td>$R_{p0.2}$ - Yield strength</td>
<td>$t_n^0$ - Normal strength</td>
</tr>
<tr>
<td>290 MPa</td>
<td>114 MPa</td>
</tr>
<tr>
<td>$R_m$ - Ultimate normal strength</td>
<td>$t_s^0$ - Shear strength</td>
</tr>
<tr>
<td>395 MPa</td>
<td>68 MPa</td>
</tr>
<tr>
<td>$A_5$ - Ultimate normal strain</td>
<td>$G_n^C$ - Normal fracture energy</td>
</tr>
<tr>
<td>12%</td>
<td>1.4 $\frac{kJ}{m^2}$</td>
</tr>
<tr>
<td></td>
<td>$G_s^C$ - Shear fracture energy</td>
</tr>
<tr>
<td></td>
<td>2.8 $\frac{kJ}{m^2}$</td>
</tr>
</tbody>
</table>

Table 1. Material data used for the SLJ [14, 15]
\[ \varepsilon_{\text{max}} = \frac{\varepsilon_n}{2} + \sqrt{\left(\frac{\varepsilon_n}{2}\right)^2 + \left(\frac{\varepsilon_s}{2}\right)^2}, \]  

in which \( \Delta D \) is the incremental damage accumulation (or incremental degradation), \( \Delta N \) is the cycle increment, \( \varepsilon_{\text{max}} \) is the principle strain and \( \varepsilon_{\text{th}} \) is the threshold strain. If the principle strain \( \varepsilon_{\text{max}} \) is below the threshold strain \( \varepsilon_{\text{th}} \), no damage accumulation occurs. The parameters \( \varepsilon_{\text{th}}, \alpha \) and \( \beta \) are calibrated based on experimental results by Khoramishad et al. [14] and are listed in Tab. 2. It shall be noted that the parameters are material dependent as well geometry and load dependent.

In order to implement the proposed fatigue model for degradation of cohesive elements (representing the adhesive), an algorithm was implemented in ABAQUS by using a USDFLD subroutine. By such an algorithm, one can attribute certain field variables to the material integration point of each element. The field variable can be updated for each increment of the FE analysis. A field variable is assigned to the tripping tractions, elastic modulus of the adhesive and also to the fracture energies of the adhesive. The subroutine GETVRM is used to determine the strains inside the cohesive elements.

As it is shown in Fig. 4, the fatigue model includes two different steps. In the first step, the maximum fatigue load is applied (under static loading conditions without damage). In this condition, the principle strains of the cohesive elements can be calculated by using GETVRM function. In the second step, the damage model is introduced to integration point of cohesive

<table>
<thead>
<tr>
<th>Epoxy FM 73M OST [14]</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
</tr>
<tr>
<td>( \beta )</td>
</tr>
<tr>
<td>( \varepsilon_{\text{th}} )</td>
</tr>
</tbody>
</table>

Table 2. Fatigue properties calibrated on test results [14].
elements based on Eq. (5) for each $\Delta N$ increment. As the damage variable ($\Delta D$) increases, the values of tripping tractions and fracture energies as well as modules of elasticity are reduced linearly. This process continues until the structure cannot stand further loading. An increment of 100 cycles per step is selected for $\Delta N$ based on a parametric study.

4. RESULTS AND DISCUSSION

The numerical model is verified based on experimental results by Khoramishad et al. [14]. The load deflection curve for a static load case is shown in Fig. 5 (left). The maximum load within the numerical model is 10.3 kN, which is 3% larger than the average test load of 10 kN. Fig. 5 (right) illustrates the SN curve of the numerical model. The points within this diagram represent test results achieved by reference test data. In general, a good agreement between numerical model and experimental results is present.

This contribution considers a fatigue load of 4 kN for further investigation. In this case, a total life time of 200000 load cycles is achieved. Fig. 6 (left) shows the back face strain curve

Figure 4. Incremental fatigue model for the cohesive elements (left) introduces linear degradation of the material properties (right) [14]
Figure 6. Back face strain curve for varying number of load cycles (left) and strain at the former zero strain point over number of load cycles (right)

for varying load cycles. For \( N = 1 \) (static load case), zero strain is present at the position \( x = 25.6 \) mm. A visible shift within back face strain occurs already after \( N = 50000 \) load cycles. At this load step, the edges are already damaged due to fatigue and thus a shift in the position of the zero strain point occurs. Strain at the former zero strain point is approximately 90\( \mu \)m/m. Fig. 6 (right) plots strain at the former zero strain point over the number of load cycles. Therefore, the results are evaluated for every 10000 load cycles. Within the first 20000 load cycles, the deviation of strain at the former neutral strain point is small. However, a distinct deviation is present at approximately 30000 load cycles. Afterwards, an increased bending of the adherent is leading to high compressive strain at the former neutral strain point. At approximately 170000 load cycles the amount of strain reaches a plateau.

In order to assess the damage within the model, the degradation of the cohesive elements is analysed. If the degradation \( D < 1.0 \), the element is degraded but not completely separated. Only in case of a complete degradation \( D = 1.0 \), presence of a crack is assumed. The smallest crack size within the model is 0.2 mm. This refers to the element size. Fig. 7 (left) compares the strain at the former zero strain point to the crack length. The first crack with a length of 0.2 mm leads to a deviation of strain of 29 \( \mu \)m/m. For increasing crack lengths up to a crack length of 4 mm, the deviation of strain is increasing almost linearly. Therefore, damage assessment is very simple for this regime. In case of a crack length of 4 mm, a large deviation of strain (approximately 800 \( \mu \)m/m) is present. At a crack length of approximately 5.2 mm, the deviation of strain reaches a plateau. From this moment on, cracks of different sizes cannot be distinguished.

Fig. 7 (right) plots the residual load carrying capabilities over the crack length. For small crack lengths, the residual strength decreases slightly. Up to a crack of approx. 8 mm, the relationship between residual strength and crack length is almost linear. For larger cracks, their is a significant drop in residual strength. In case of a crack of 12 mm, residual strength drops
below 4 kN. This refers to a critical crack which leads to failure under fatigue loading.

Monitoring strain at the former zero strain position easily detects cracks within the adhesive up to an crack length of 6 mm. Already a very small crack length of 0.2 mm can be detected. If the crack exceeds 6 mm, the structure still has enough load carrying capabilities in order to withstand fatigue load. However, crack propagation cannot be monitored for such cracks. Therefore, the structures integrity has to be insured with alternative measures. In this case, a repair action is recommended. However, the operator might also consider alternative monitoring methods (e.g. regular scheduled visual inspections) in order to ensure structural integrity while expanding life time of the adhesive joint.

The results indicated a relationship between crack length and strain at the former zero strain point as well as residual strength. This relationship enables an intrinsic damage assessment. Certain values of measured strain refers to particular crack length and hence to particular residual strength. Using this simple relationship, an automatic damage assessment can be used. If the measured value exceeds a predefined threshold, which refers to crack length of 6 mm, a warning signal will be sent to the operator. Fig. 8 visualises the relationship between strain at the former zero strain point and the residual strength. The two different domains are easily seen within the trend of the curve. For smaller crack lengths, the relationship is almost linear. For a crack larger than 6 mm, the method is not able to distinguish between different crack lengths and therefore fails to predict the remaining load carrying capabilities. Due to the linear relationship for small cracks, the residual strength is approximated by

\[ F_{max} = -1663500 \text{ N} \cdot \varepsilon_{xx}(x = 25.6 \text{ mm}) + 10203 \text{ N}. \]  

(7)
5. CONCLUSION

This contribution demonstrates the use of SDIs for SLJ. Based on structural analysis, a zero strain point of the back face strain curve is derived as SDI. This zero strain point provides an optimised sensitivity for cracks within the adhesive. In order to verify that approach, the SLJ is modelled in FEM including a fatigue modelling of the adhesive[14].

The FEM results indicate, that an increasing amount of load cycles leads to a shift of the zero strain point. Therefore, the former zero strain point is exposed to strain. For cracks up to 6 mm, a linear relationship between deviation of strain at the former zero strain point and the crack length is present, as well as between the crack length and the remaining load carrying capabilities. For larger cracks, the approach is not able to distinguish between different crack lengths. However, such large cracks refer to later stages of the fatigue life time. The advantage of the proposed SDI is his sensitivity against small cracks within the adhesive. Already cracks of a length of 0.2 mm might be detected. Afterwards, there is long period where crack propagation might be monitored. Therefore, there is plenty of time for scheduling maintenance after damage detection. In addition, an intrinsic damage assessment is applicable, due to the relationship between measured value and the structural behaviour of the SLJ. Within intrinsic damage assessment, the residual strength is derived by the measured strain using a linear approximation.

The proposed structural damage indicators utilise strain of a distinct point. However, common strain sensors cover a certain (small) area, where an average strain is measured. The influence of the extent of the strain sensor is supposed to be addressed in future work. This contribution focusses on fatigue cracks at the edges of a SLJ. However, adhesive joints are prone to manufacturing defects, namely voids and weak bonds. Those uncertainties are addressed in future work as well.
References


DISCRETE DAMAGE MODELING OF INITIATION AND PROPAGATION OF MATRIX CRACKING AND DELAMINATION IN CLAMPED TAPERED LAMINATED BEAM SPECIMENS

Hari K. Adluru*, Kevin H. Hoos*, Endel V. Iarve*,†

*IPPM, University of Texas at Arlington Research Institute
7300 J Newell Blvd., Fort Worth, TX 76118
Hari.Adluru@uta.edu

†University of Texas at Arlington
500 W. First Street, WH 300A, Arlington, TX, 76019-0018
Endel.Iarve@uta.edu

Key words: Cracking, Delamination, Regularized Extended FE

Summary: The Discrete Damage Modeling (DDM) method was used for failure initiation and progression analysis in tapered composite beam. DDM employs Regularized Extended Finite Element Method (Rx-FEM) which allows modeling the displacement discontinuity associated with individual matrix cracks in individual plies of a composite without regard to mesh orientation by inserting additional degrees of freedom in the process of the simulation. Cohesive zone method is used for propagation of the mesh independent cracks and delaminations. The tapered composite beam specimen allows studying the matrix crack initiation from pristine condition as well as delamination initiation and evolution including migration from one interface to another. The predicted results showed good agreement with experimental data for a number of key metrics such as peak loads and delamination migration distances. Such agreement was observed when using ply level transverse strength value obtained by using three point bend test method, whereas the results obtained by using a lower value resulting from tensile testing of 90° coupons resulted in under-prediction of the peak load.

1 INTRODUCTION

Laminated composite materials are used in stiffness critical applications and contain various stress concentration features such as holes, tapers and stiffener attachments. It is these areas which often determine the load carrying capacity of the structural elements and thickness scaling of critical parts. A representative subelement is studied in this work with the aim of evaluating the ability to predict the mechanical response of such critical regions by using discrete damage modeling. Clamped Tapered Beam Specimen (CTBS) designed by Advanced Composite Project Team at NASA Langley Research Center (LaRC) in Hampton, VA is considered [1]. This specimen allows studying the matrix crack initiation from pristine condition as well as delamination initiation and evolution including migration from one interface to another. The clamped tapered beam specimen is an evolution of the delamination migration specimen proposed by Ratcliffe et al. [2]. The main difference in CTBS is the
tapered edge of the short arm and the absence of initial delamination, thus allowing one to study not only the delamination migration phenomena but also the failure initiation from pristine tapered edges. We will focus on failure initiation in the form of matrix cracking, its transition to the delamination failure mode and subsequently the migration of the delamination from one interface to another.

The predictions were performed simultaneously with experimentation conducted at NASA LaRC. The first round of predictions was performed prior to knowing the experimental data and the second round of predictions was performed after the experimental data was revealed. The parameters of interest included the peak load as well as delamination migration location and load.

We will begin by discussing the DDM analysis methodology followed by problem statement and CTBS description and conclude by discussing the blind and corrected predictions and results comparison.

2 MESH INDEPENDENT CRACK (MIC) MODELING BY RX-FEM APPROACH

The DDM approach consists of mesh-independent modeling of matrix cracks in each ply of the laminate, and modeling the delamination between the plies by using a cohesive formulation at the ply interface. The matrix cracks are modeled by using the Regularized eXtended Finite Element (Rx-FEM) formulation [3]-[5]. This formulation is a derivative of the original x-FEM proposed by Moes et al. [6], where the cracked element is enriched by additional degrees of freedom to ensure the displacement jump across the crack face. These additional degrees of freedom are associated with the shape functions, which are essentially partitioned along the crack surface, and represented by the Heaviside step function. The regularized formulation replaces the Heaviside step function with continuous function changing from 0 to 1 over a narrow volume of the so called gradient zone. In our approach, the step function is approximated by the same shape functions as the displacements. In this case, the Gauss integration points of the initial approximation may be used for integration of the enriched functions. This also simplifies the interaction between plies with different crack orientations because they maintain a common integration schema. The formalism tying the volume integrals in the gradient zone to surface integrals in the limit of mesh refinement was discussed by Iarve et al. [3].

A critical aspect of the regularized formulation is the constitutive behavior in the gradient zone. The approach taken below directly incorporates the surface discontinuity-based cohesive law developed by Turon et al. [7] in the formulation of the fracture energy of the gradient zone. The cohesive force \( \tau \) resisting the opening displacement jump \( \Delta u \) at an arbitrary crack surface point is:

\[
\tau = (1 - d)K\Delta u + dK\langle \Delta u \rangle n
\]

(1)

where \( K \) is a high initial stiffness and \( d \) is the damage parameter. The first term in Eqn. (1) represents the crack cohesive force, and the second term prevents interpenetration of the crack surfaces. The brackets \( \langle x \rangle = \frac{1}{2}(x + |x|) \) represent the McAuley operator and vector \( n \) is the unit normal vector to the crack surface. The damage parameter controls the crack opening, and will be used below to display the length of the cracks. A bilinear relationship between the absolute values of \( \tau \) and \( \Delta u \) is assumed and defined by the initial and final values of the displacement gap \( \Delta_0 \) and \( \Delta_f \). The initial value of the gap corresponds to the onset of
bond softening, so that $d=0$ if $\Delta u \leq \Delta_0$ and the final value $\Delta_f$ of the displacement gap corresponds to complete separation, i.e. $d=1$ if $\Delta u \geq \Delta_f$. The typical ratio of $\Delta_f/\Delta_0 = 2K G_c/X^2$ is on the order of magnitude of $10^4$ due to very high values of the initial bond stiffness $K$, where $G_c$ and $X$ are the critical value of the Energy Release Rate (ERR) and the initial strength respectively (such that $\Delta_0 = X/K$). The value of the damage variable $d=0.5$ corresponds to a very small displacement gap of approximately $2\Delta_0$ whereas the displacement gap of approximately $0.5\Delta_f$, which is indicative of interface separation, corresponds to a damage variable value of $d=1-\Delta_0/\Delta_f$, which is very close to 1. In the results section the figures displaying the matrix cracking and delamination path correspond to the CZM with $d \geq 0.995$. The same cohesive law is also used to model the delaminations between plies.

3 SOLUTION ALGORITHM

A flow chart of a typical simulation is shown on Figure 1. A simulation begins without any initial matrix cracks. At each load level a failure criterion is used to find the location and orientation of the matrix crack cohesive zone to be inserted. In the present paper the 3D LaRC04 criterion [8] is used. The criterion is evaluated at each integration point and, if the criterion is exceeded a cohesive zone corresponding to MIC is inserted by using Rx-FEM without regard to mesh orientation. This matrix crack, i.e. the MIC cohesive zone is oriented in the fiber direction under an angle to the ply surface, which is calculated by LaRC04 failure criterion. LaRC04 predicts the crack angle with respect to ply surface, which is $90^\circ$ under tensile normal stresses whereas in shear and compression dominated loading states it can significantly differ from $90^\circ$. Note that the MIC cohesive zone is closed at insertion and has no effect on laminate response. In the next loading increment this cohesive zone begins to open and forming a matrix crack. At the core of the algorithm is a non-linear Newton-Raphson (NR) solution loop, which equilibrates the displacement field at a given load level by iteratively finding the distribution of the damage variables of interface and MIC cohesive laws in all integration points. After the convergence criterion is met the step is finished and the new crack insertion procedure is performed. Here and below we will be using the term matrix crack interchangeably with the MIC cohesive

Figure 1. Flow Chart of the Automatic Mesh Independent Crack (MIC) Insertion and Growth Procedure
zone in the modeling context. Note that within the described algorithm new cracks cannot be inserted prior to when the equilibrium is established, which had important implications for modeling the CTBS. Indeed, as will be seen below, we over-predicted the delamination migration distance when such migration happened dynamically in the experiment during the unstable propagation, while the model only allowed for migration path when the unstable propagation stopped.

4 CLAMPED TAPERED BEAM SPECIMEN (CTBS)

The schematics and dimensions of the specimen are shown on Figure 2. The ply drops are arranged to form a 30° ramp and a small 3.175 mm (0.125 in) radius cusp at the transition to the skin at the edge-of-flange. The skin is made of 14 ply, 190 gm/m² IM7/8552 Carbon Fiber Reinforced Plastic (CFRP) and the flange is also a 14 ply, IM7/8552 cross ply laminate. Each ply was approximately 0.18mm thick. The plies in the laminate layups are listed from the bottom to the top [0/903/0/902/90/0/902/0/90] in the skin and [903/02/90/902/90/902/903] in the flange respectively. As a result, a 4 ply 90° stack is formed between the upper 0° ply of the skin and bottom 0° ply of the flange. The specimen is clamped on the right and left edge regions to a length of 31.5 mm (1.24 in).

The boundary conditions were applied in the edge regions marked on Figure 2 and consisted of constraining of vertical $u_z$ displacement on the left side and both $u_x$ and $u_z$ on the right side edge. Since only 3D hexahedral elements are available for Rx-FEM simulation in BSAM a 3D model was created. Only one element in the y –direction was used taking advantage of the 2D nature of the problem and the $u_y=0$ condition was imposed at the $y=0$ plane, thus providing a half width symmetric model. Three different meshes were created in Abaqus™ CAE 2016 and imported to BSAM software [9], where the Rx-FEM is implemented for fracture modeling. Coarse model has approximate element size of 0.18 mm, Fine model has an approximate element size of 0.09 mm and Finer model has an approximate element size of 0.045 mm. These element sizes correspond to approximately 1, ½ and ¼ of the ply thickness. It is anticipated that matrix cracks will form in the 4 ply 90° stack.
between the skin and the flange. The $90^\circ$ plies in this region and plies immediately above and below are the main focus regions. The coarse mesh is practically uniform through the specimen. The $1/2$ and $1/4$ ply meshes were locally refined in and around the $90^\circ$ ply stack

Table 1. Unidirectional im7/8552 ply stiffness and strength properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{11}$ (GPa)</td>
<td>157.2</td>
</tr>
<tr>
<td>$E_{22}, E_{33}$ (GPa)</td>
<td>8.960</td>
</tr>
<tr>
<td>$G_{12}, G_{13}$ (GPa)</td>
<td>5.08</td>
</tr>
<tr>
<td>$G_{23}$ (GPa)</td>
<td>2.99</td>
</tr>
<tr>
<td>$n_{12}, n_{13}$</td>
<td>0.32</td>
</tr>
<tr>
<td>$n_{23}$</td>
<td>0.5</td>
</tr>
<tr>
<td>$a_1$</td>
<td>0.00</td>
</tr>
<tr>
<td>$a_2$ ($1/C$)</td>
<td>3.01e-05</td>
</tr>
<tr>
<td>$Y_T$ (MPa)</td>
<td>64 and 127</td>
</tr>
<tr>
<td>$Y_C$ (MPa)</td>
<td>200</td>
</tr>
<tr>
<td>$S$ (MPa)</td>
<td>95.8</td>
</tr>
<tr>
<td>$X_T$ (Gpa)</td>
<td>2.723</td>
</tr>
<tr>
<td>$X_C$ (Gpa)</td>
<td>1.2</td>
</tr>
<tr>
<td>$G_{IC}$ (N/mm)</td>
<td>0.24</td>
</tr>
<tr>
<td>$G_{IC}$ (N/mm)</td>
<td>0.739</td>
</tr>
</tbody>
</table>
between the skin and flange. The three meshes in the tapered area are shown on Figure 3. The material properties listed in Table 1 were provided by NASA LaRC and include both elastic stiffness and strength properties needed for modeling failure initiation and propagation. The material system IM7/8552 is well characterized by using standard [10] and custom testing methodologies [11]-[13]. A particular property, namely the transverse tensile strength, appears to be of special importance for predicting the initiation of failure in CTBS. The sensitivity of this property to the method of measurement is also well known and can result in the difference of as much as two times comparing the values obtained from standard ASTM D3039 tensile testing of the 90° coupons [10] and 3 point bend testing according to ASTM D790, Ref. [13], [14]. Thus the blind predictions were made for two values of transverse tensile strength of $Y_t=64\text{MPa}$ and $Y_t=127\text{MPa}$.

5 RESULTS AND DISCUSSION

5.1 Initial (blind) Predictions

The clamped boundary conditions imposed in the experiment represent a modeling challenge since a certain degree of slip and friction which is present in the clamped region makes it difficult to match the measured stiffness. Thus prior to prediction the experimental data allowing the stiffness calibration was provided. Digital Image Correlation measurement of the displacement distribution along the specimen length at several load levels was performed and provided for both configurations. The results of predicted deflection and that measured on 3 different specimens are shown on Figure 4. The experimental data for individual specimens are shown by dashed line and the prediction by solid line. Despite a good agreement additional tabulated stiffness data indicated that the modeled specimen is slightly stiffer. The predicted stiffness is 496 N/mm and the stiffness of the experimental samples are between 450 N/mm and 464 N/mm. The initial blind predictions were performed with the coarse mesh. The predicted and measured load vs. displacement data are shown on Figure 5. As expected the transverse strength $Y_t$ has a significant influence on the predicted maximum load. The low value of 64MPa resulted in 30-40% under prediction of the specimen strength whereas the high value of the transverse strength provided good correlation with experimental data. It became apparent that the subsequent refinements and mesh sensitivity studies should be performed with the $Y_t=127\text{MPa}$. In the next section we will report in detail the modeling results performed mostly after the experimental data was known.
5.1 Correction Stage Simulations and Failure Process Details

In this section we will describe results obtained after the experimental data was known with exception of the coarse mesh results which were obtained in the blind prediction phase. There were no model changes other than mesh refinement performed on the second phase of the analysis. Several key events such as MIC CZM insertion load and displacement; peak force and displacement and delamination migration distance and the applied force were
documented. Force displacement plots for different mesh sizes are shown on Figure 6. The list of key events are recorded in the Table 2. As the mesh size becomes finer the insertion of the MIC CZM occurs earlier and is very sensitive to mesh refinement. It is attributed to the specimen design where the critical location exhibits a weak stress singularity typical to a ply interface and free edge intersection [15]. In this case the mesh refinement will always result in higher maximum stress value at any given applied load and trigger the failure criterion sooner. Nevertheless the peak load corresponding to the propagation of MIC is practically the same for the ½ and ¼ ply meshes. Such mesh independence is a result of cohesive law formulation where the failure can initiate early but propagation occurs only when the stored energy is sufficient for fracture. The distance from the location of delamination migration to the loading hinge is shown on Figure 7. The delamination migration in the present model is a sequence of several events. First the delamination grows from the initial matrix crack to certain length at which it is in the equilibrium at given load. Next the MIC CZM is inserted at the integration point with highest stress, which in this case is most likely at the delamination front. It is possible that at the new load step the MIC will not open and the delamination continues to propagate or the MIC opens and extends through the thickness of the ply stack. At this stage the original delamination may stop growing and a new delamination will start growing from the crack tip on the other interface resulting in clean migration as on Figure 7.a. In another scenario the original delamination will continue growing and spinning off cracks as on Figure 7.b until one MIC opens completely and migration occurs. Such process requires additional load as is the case with Fine mesh prediction on Figure 6. Yet a different migration pattern on Figure 7.c involves only one crack, however, the original delamination still grows a certain length past the root of the crack before the new delamination started to propagate. Refining the mesh led to closer agreement with the experimentally observed range for

![Figure 6 Force displacement graph for different mesh sizes.](image)

Crack

Coarse Mesh

Fine Mesh

Finer Mesh

0.00 0.50 1.00 1.50 2.00 2.50 3.00 3.50

Displacement (in mm)

0 100 200 300 400 500 600

Force (in N)
<table>
<thead>
<tr>
<th>Description</th>
<th>Migration distance ($a_m$ in mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse model</td>
<td>18.5</td>
</tr>
<tr>
<td>Fine model</td>
<td>15.2</td>
</tr>
<tr>
<td>Finer model</td>
<td>12.3</td>
</tr>
<tr>
<td>Experimental</td>
<td>8.13-13.49</td>
</tr>
</tbody>
</table>

Table 3 migration distance from load application.

Figure 7 Crack migration distance from load application a) 1 ply mesh b) ½ ply mesh c) ¼ ply mesh
delamination propagation distances. The common reason for over predicting the migration distance was unstable crack propagation past the experimentally observed migration location. Mesh refinement clearly appeared to result in more stable crack propagation thus bringing the predictions into the experimental range.

6 CONCLUSIONS

The Rx-FEM was applied to prediction of matrix crack and delamination initiation and propagation including its migration from one interface to another. The simulations were performed simultaneously with experimental measurements conducted by Dr. James Ratcliffe at NASA LaRC and showed good quantitative agreement with experimental data in most cases. This conclusion was based on comparing a number of key metrics such as peak loads and delamination migration distances. Further work will be conducted to understand the implications of the specimen variation on the prediction accuracy as well as simulation of the delamination migration in the unstable propagation conditions.

7 ACKNOWLEDGMENTS

This work was funded under NASA contact number NNL16AA02C with the University of Texas at Arlington, Arlington, TX. The authors are grateful to Drs James G. Ratcliffe, Nelson Carvalho, Ron Kruger, Kevin O’Brien and the Advanced Composite Consortium members for encouragement and fruitful discussions.

REFERENCES


A FATIGUE DAMAGE MODEL FOR FIBER METAL LAMINATES INCLUDING FIBER/MATRIX DEGRADATION

C. Gerendt†, E. Jansen, R. Rolfes

Leibniz Universität Hannover, Hannover, Germany
Institute of Structural Analysis (ISD)
Appelstraße 9A, 30167 Hannover
†c.gerendt@isd.uni-hannover.de

Key words: Fiber metal laminates, FE analysis, Fatigue crack growth, Fatigue damage

Summary: A finite element based fatigue damage model for fiber metal laminates is presented. The model is primarily characterized by the capability to account for possible fatigue damage processes in the fiber-reinforced plastic layers of the laminate. Furthermore, a crack growth approach based on plastic strain energy is applied, which is beneficial in several aspects compared to elastic fracture mechanics approaches, especially in case of variable amplitude loading. A numerical example is presented, which shows the sensitivity of the crack growth rate in the metal layers concerning the state of damage in the fiber-reinforced plastic layers.

1 INTRODUCTION

Today’s fiber metal laminates (FMLs) originate from metal laminates (adhesively bonded layers of sheet metal), which were developed mainly for aircraft applications starting in the 1950s. Metal laminates already featured an excellent fatigue resistance and damage tolerance in comparison to monolithic metals, respectively. Later glass, aramid or carbon fibers were added to the adhesive in order to reduce the risk of through cracks (simultaneous crack growth in all sheets of a metal laminate) [1]. Since the fibers generally tend to be more fatigue resistant than the metal layers, possible cracks in the metal layers can be bridged by the intact fibers (see Fig. 1), which reduces significantly the crack growth processes in FMLs, resulting in an excellent damage tolerance of the laminate. The efficiency of the fiber bridging is thereby, amongst others (for example stiffness and strength of the bridging fibers), influenced by the amount of delamination, which occurs due to the load transfer from the cracked metal to the intact fiber-reinforced plastic (FRP) layers. With increasing delamination the stresses in the fibers decrease, which reduces the bridging efficiency so the crack growth in the metal layers accelerates. A schematic illustration of this fiber bridging effect is given in Fig. 1.

Since also the condition of the bridging fibers (or in general of the bridging FRP layer) is naturally essential for the efficiency of the crack bridging effect, the fatigue damage of the FRP layers should be taken into account when simulating the fatigue damage of FMLs. Although much effort has been put into the development of sophisticated (mostly analytical) fatigue damage models for FMLs during the past decades, none of the established models takes into account possible FRP damage due to cyclic loading. Only in [17] a first attempt to account for FRP damage within a FML fatigue damage model is presented, whereby a quite close fit to corresponding experimental crack growth rates could be obtained. Furthermore, attention has been barely paid so far to the elasto-plastic character of the metal layers concerning the fatigue damage analysis, since many of the established models [4] are formulated on an analytical
basis (and are therefore not capable of taking into account any plastic material response) or apply linear-elastic fracture mechanics to account for possible fatigue crack growth. To assume a purely elastic response of the metal layers especially complicates the analysis of variable amplitude loading scenarios, where load sequence or crack closure effects must be taken into account [3]. A recent summary of the most established models for fatigue damage analysis of FMLs is given in [4].

In this work a first version of a finite element (FE) based fatigue damage model for FMLs is presented, which was developed to overcome the aforementioned shortcomings of today’s established analytical FML fatigue damage approaches [4]. The herein presented framework accounts for possible fatigue damage in the FRP layers as well as for the elasto-plastic material behavior of the metal layers under cyclic loading. Detailed crack modeling allows furthermore to account for crack closure effects under compressive loading.

2 THEORETICAL BASIS OF THE FATIGUE DAMAGE MODEL

The fatigue damage model that is presented in this work has the innovative characteristic to account for possible fatigue damage in the FRP layers of the FML by applying an energy based fatigue damage model recently presented by Krüger [5]. Furthermore, the crack propagation in the metal layers is not predicted by using linear-elastic fracture mechanics methods, but by adapting a rather innovative elasto-plastic approach firstly presented in [6] and later validated by Nittur [7], based on findings of Klingbeil [8]. The aim during the development of the presented model was to create a FE-based framework for fatigue damage analysis of FMLs, capable of analyzing the fatigue damage of real structural components under realistic loading scenarios (e.g. overloads, load sequences). By applying non-empirical fatigue damage approaches for the laminate constituents, the model is applicable in an early design phase of a FML.

2.1 Crack growth in the metal layers

The approach applied to predict the crack growth in the metal layers is based on assumptions...
made by Bodner [9], who stated that the fatigue crack growth in metallic structures is mainly caused by plastic strain energy that is accumulated within a small area ahead of the crack tip, the so-called process zone. Klingbeil [8] showed that it is possible to calculate this energy accumulation using FE-analysis, where the reversed plastic zone (RPZ, see Fig. 2) is interpreted as process zone. A sophisticated numerical framework to apply the plastic energy accumulated in the RPZ for a FE-based crack growth model was presented in [6] and later on modified and validated by Nittur [7]. The Abaqus/Python based approach presented in [7] inspired the development of a new framework based solely on Abaqus subroutines to account for fatigue crack growth in the metal layers of the FML.

The actual procedure of the crack growth analysis used in this work is schematically illustrated in Fig. 3. The algorithm can be used in two different configurations, which will be denoted with **jumps on** and **jumps off** in the following (see Fig. 3). The configuration **jumps on** can be described as follows: first of all the plastic energy increment in the RPZ $\Delta W_{RPZ}^P$ throughout one simulated load reversal for a given crack length is determined within the UMAT subroutine. After the energy increment has been determined numerically, the energy based crack propagation criterion is evaluated. If the threshold value $W_{crit}^P$ is exceeded, the crack will extend by one or more ($N_{el}$) elements, depending on how much the critical value has been surpassed. If the critical value is not exceeded immediately after one load reversal, the load reversals that are necessary to exceed the critical value with the given energy increment $\Delta W_{RPZ}^P$ are computed within the UMAT subroutine. The basic assumption of this ‘cycle jump technique’ is that in case of constant amplitude loading the energy increment $\Delta W_{RPZ}^P$ is nearly constant for a certain crack length. By determining the energy increment only once for a certain crack length, the efficiency of the analysis can be increased significantly compared to the **jumps off** configuration, which is quite similar to the approach presented in [7]. Since in the **jumps off** configuration each load cycle is simulated in order to obtain the energy increment, the

![Figure 3: Illustration framework to simulate the crack propagation in the metal layers. The different procedure configurations (**jumps on, jumps off**) are to be chosen by the user.](image)
computational costs increase clearly. Nevertheless, the jumps off configuration is advantageous in case of variable amplitude loading, where the energy increment changes significantly from cycle to cycle, even without any crack extension. However, both configurations of the presented crack growth algorithm could be verified successfully with the results presented in [7] and were also validated by experimental crack growth rates, see Section 3.1.

To obtain a mesh independency of the applied energy criterion, the threshold value $W_{\text{crit}}^P$ is scaled according to the element size in the process zone: the value increases with increasing element size and decreases for smaller elements. Accordingly, to propagate the crack through a large element (large crack increment) more energy has to be dissipated than for a smaller element. The energy threshold value is furthermore assumed to be a material parameter, which yet has to be identified by model calibration. Accordingly, the threshold value is adjusted in such a way that the model yields similar crack growth rates as observed experimentally. In Section 3.1 the model has been successfully calibrated for aluminum 2024-T3.

The crack propagation of $\Delta a$ [mm] is modeled using a subroutine-based node release technique. Here, the subroutine for user defined multiple point constraints (MPCs) is adapted, whereat the command for node release is given by the crack growth analysis in the UMAT subroutine. Contact definitions (hard normal, frictionless tangential) are furthermore defined along the crack flanks in advance to obtain kinematically sound results in case of crack closure.

2.2 Stiffness / strength degradation in the FRP Layers

The fatigue damage analysis of the FRP layers is performed by adaptation of a physically based fatigue damage model for FRPs presented by Krüger [5]. The model has the advantage that laminates can be evaluated layer-wise concerning fatigue damage processes, while the application of the Puck failure criterion offers further information concerning the particular failure mode related to each material point (integration point) in the laminate. For the current

Figure 4: Schematic illustration of the damage model used for determination of the FRP degradation parameters $\eta_E$ (stiffness) and $\eta_R$ (strength) according to [5]
C. Gerendt, E. Jansen, R. Rolfes

version of the FML damage model presented in this work, the failure modes ‘fiber failure’ and ‘inter-fiber failure’ for a tension-tension loading were implemented according to [5]. Since several types of GLARE usually contain only 0- and 90-degree layers [10], these two intralaminar damage modes were considered as particularly relevant for the presented FML damage model.

The theoretical basis of the FRP damage model [5] is the hypothesis of Pfanner [11], who stated that the damage state (stiffness and/or strength degradation) of a quasi-statically loaded material and that of a cyclically loaded material are comparable, if the amount of dissipated energy is equal. This means, that a cyclically applied loading will yield the same state of material damage as a quasi-statically applied loading, as long as the dissipated energy is the same for both cases. Knowing the quasi-static stress-strain curve of the material, the damage due to cyclic loading can therefore be determined by setting the cyclically dissipated energy equal to the corresponding quasi-static energy (area under the stress-strain curve). By doing so, the stiffness and strength degradation parameters necessary to fit the cyclic energy under the quasi-static stress-strain curve (static energy) are obtained iteratively. The aforementioned methodology is illustrated in Fig. 4. The implemented FRP damage analysis will be validated in Section 3.2.

2.3 Procedure of the overall model

In order to develop the fundamental framework for the presented FML fatigue damage model, the primary task was to merge the two methods presented in Section 2.1 (crack growth in the metal layers) and 2.2 (degradation of the fiber/matrix material) appropriately. The workflow of the FML damage model is illustrated in Fig. 5 and will be described in the following.

The description of the model framework starts in an arbitrary ‘cyclic step’. Beside the energy accumulation that is performed throughout the step (see Section 2.1), the maximum and

![Workflow illustration of the presented fatigue damage model for FMLs](image-url)
minimum stress values in the FRP material are monitored for the subsequent FRP degradation analysis (see Section 2.2). After a cyclic step is finished, the crack growth analysis in the metal layers is performed, using the jumps on configuration presented in Section 2.1, Fig. 3. Several parameters are obtained by the crack growth analysis: first of all the crack growth rate $\Delta a/\Delta N$ [mm/cycle] is determined, which is essential to evaluate the fatigue behavior in terms of fracture mechanics. The evaluation of the energy criterion gives furthermore the crack growth increment $\Delta a$ [mm]. The crack growth rate $\Delta a/\Delta N$ in combination with crack growth increment $\Delta a$ yield the number of cycles $\Delta n$ required to elongate the current crack by $\Delta a$. After convergence of the crack growth analysis the values of $\Delta a$ and $\Delta n$ are passed to the FRP damage algorithm and the MPC subroutine, which are both initiated during the so-called ‘flatline step’, following each cyclic step, see Fig. 5.

The FRP damage analysis (see Section 2.2) is performed using the maximum/minimum stress values gathered in the previous cyclic step as well as the number of cycles $\Delta n$ obtained by the crack growth analysis. The node release is realized using the MPC subroutine available for Abaqus. The number of nodes that have to be released is defined by the crack growth increment $\Delta a$ that is transferred to the MPC subroutine from the previous crack growth analysis. It turned out, that the node release might lead to massive convergence issues, mainly due to the predefined contact formulations along the crack flanks that emerge rather suddenly as soon a node is released. This problem can be solved by introducing a sufficient amount of contact damping to the model during each of the flatline steps, whereby the damping is ramped down throughout the flatline step and inactive in the following cyclic step. This node release technique offers the ability to model crack growth without the usage of any basic redefinitions of the actual model, e.g. using an additional Python routine (see [7]).

Some first results obtained with the developed FML fatigue damage model are presented in Section 3.3. The presented results demonstrate that the degradation of the FRP is able to influence significantly the crack growth rate in the metal layers.

3 NUMERICAL RESULTS: VERIFICATIONS AND VALIDATIONS

In the subsequent sections some numerical results obtained by the methods described in Section 2 are presented. In Section 3.1 the crack growth algorithm is verified based on numerical results presented in [7]. Additionally, the calibration of the crack growth model is performed for aluminum 2024-T3 sheet metal, in order to obtain the energy threshold value $W_{CIT}^P$. In Section 3.2 the strength degradation obtained by the implemented FRP damage model is validated. In Section 3.3 some first results obtained by the developed FML fatigue damage model are presented, which indicate, that damage processes in the FRP layers can influence the crack growth in the metal layers significantly.

3.1 Verification and calibration of the crack growth model

In order to verify the crack growth algorithm described in Section 2.1 numerical results and material properties documented in [7] were utilized. Fig. 6 shows that the adaptation of the Abaqus/Python based framework from [7] could be successfully achieved for both configurations (jumps on, jumps off, see Fig. 3) of the developed Abaqus/Subroutine based framework. Furthermore, the similarity of the results obtained by the different configurations supports the validity of the proposed cycle jump thesis, which is that $\Delta W_{RPZ}^P$ can be assumed as constant for a given crack length under constant amplitude loading. Fig. 7 shows, that the
C. Gerendt, E. Jansen, R. Rolfes

jumps on configuration is accordingly able to predict correctly the experimental crack growth rates presented by Nittur [7]. The benefits that are to expect from the presented elasto-plastic crack propagation model in the jumps off configuration can be demonstrated by considering the scenario of a single overload. Fig. 8 shows, that the three characteristic stages, namely acceleration, retardation and regeneration [1], in case of a single overload are described qualitatively correctly by the model. The model also takes successfully into account the dependence of the intensity and the duration of the retardation phase concerning the overload amplitude. This relation can also be observed in corresponding experimental investigations of elasto-plastic materials [1]. Such effects are barely predictable with any approach based on linear-elastic fracture mechanics [3], generally used for FML fatigue analysis [4].

Figure 6: Verification of the crack growth model for different stress intensity ranges using the results and material properties presented in [7]. The black framed symbols are the ones originally used in [7] (NJ – jumps off, J – jumps on).

Figure 7: Comparison of crack growth rates predicted by the jumps on configuration of the developed crack growth framework with (experimental) results from [7] show the validity of the applied cycle jump technique. Material properties taken from [7].

Figure 8: Effect of a single overload concerning the crack propagation rate (results obtained with the jumps off configuration)
A detailed explanation of the determination of the energy threshold \( \Delta W_{\text{crit}} \) is given in the following. Since an analytical definition of this material parameter is still a topic of current research [13], the critical energy value \( \Delta W_{\text{crit}} \) has to be calibrated according to experimental results. Therefore, the critical energy value has to be defined in such a way that the crack growth rates resulting from the model are similar to experimentally measured growth rates for a given stress intensity range. This means, that only one point from a \( (\Delta a/\Delta N) - \Delta K \) curve (instead of the whole curve) is necessary as experimental input for the crack growth analysis. This significantly reduces the experimental effort needed for the determination of the model input parameters. For the analysis performed in Section 3.3 the crack propagation model is calibrated for aluminum 2024-T3, which is generally used for the FML GLARE [10]. The experimental basis for the calibration is taken from a NASA report [12], where the crack growth rates of aluminum 2024-T3 sheet metal were studied for different mean stress values. The predicted crack growth rates after successful calibration of the model as well as the applied material parameters are shown in Fig. 9 and Table 1, respectively.

![Figure 9: Calibration of the crack propagation model for a 2.28 mm thick aluminum 2024-T3 sheet metal](image)

Table 1: Material properties of the metal layer used for simulations in Section 3.3

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stiffness</td>
<td>71.800 MPa</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.33</td>
</tr>
<tr>
<td>Yield strength</td>
<td>353 MPa</td>
</tr>
<tr>
<td>Hardening modulus</td>
<td>(-0) MPa</td>
</tr>
<tr>
<td>Energy threshold</td>
<td>70 mJ/mm²</td>
</tr>
</tbody>
</table>

Although the calibration process already reduces the required experimental input of the model significantly compared to established fracture mechanics approaches, further research is necessary to derive the critical energy value \( \Delta W_{\text{crit}}^{P} \) on a physical basis, which could not be achieved to date [13]. A physically based derivation of this parameter would give the possibility for a crack growth model that would be nearly independent from any experimental fracture mechanical input parameters.

### 3.2 Validation of the FRP damage model

In order to validate the applied methods for fiber/matrix degradation, experimental investigations [14] considering the residual strength of cyclically loaded GFRP specimens were taken into account. The validation is performed with a simple FE-model, containing only four continuum shell (SC8R) elements. The constant stress amplitudes of the experimental tests
were applied using a stress ratio of $R = 0.1$, whereas the total number of load cycles was split into 10 intervals each. The material properties used for the simulations in Section 3.3 are listed in Table 2. The strength degradation obtained by the UMAT subroutine as well as the experimentally measured residual strength of the cyclically loaded GFRP probes are shown in Fig. 10a (longitudinal / fiber degradation) and 10b (transversal / matrix degradation). The results show that the implementation predicts the strength degradation in fiber direction (fiber dominated) and perpendicular to the fiber direction (matrix dominated) quite accurately. The validation of the material stiffness degradation will be carried out in future work.

### 3.3 Verification of the FML damage model

The FML fatigue damage model that has been described in detail in Section 2 is to be tested considering a simple ‘plate with hole’ example, see Fig. 11. The geometrical measurements were chosen referring to specimen requirements defined by the norm AITM1-0009 [15]. Since no delamination approach has been implied in the framework (Fig. 5) so far, an arbitrary

<table>
<thead>
<tr>
<th>$E_{11}$</th>
<th>$E_{22}$</th>
<th>$\nu_{12}$</th>
<th>$G_{12}$</th>
<th>$R_{11}$</th>
<th>$R_{22}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>40,000 MPa</td>
<td>14,100 MPa</td>
<td>0.278</td>
<td>6.755 MPa</td>
<td>814 MPa</td>
<td>55 MPa</td>
</tr>
</tbody>
</table>
delamination strip has been predefined, which has a width of 0.5 mm on each side of the crack path (see Fig. 11). To reduce the computational costs only a quarter of the overall model is analyzed using corresponding symmetry boundary conditions. The thickness of the layers has been chosen referring to standard measurements given in [10]. Since the increase of computational efficiency of the framework is still in the scope of current developments, the load has been chosen high enough to cause the damage processes of interest (crack growth and FRP damage) within an appropriate amount of computation time. Accordingly, a laminate load of 170 MPa with a stress ratio of $R = 0.1$ is applied.

The model (Fig. 11) consists of continuum shell elements (SC8R), which take into account a plane stress state while offering the advantages of 3-D modeling. The ability to model the actual layer thickness was considered to be beneficial especially in terms of crack growth modeling within the laminate. It is furthermore to mention, that the crack initiation phase has not been taken into account for this first verification example. Nevertheless, the initial crack length has been set to $a_i = 1$ mm, which is considered to be fundamental for obtaining the number of cycles for crack initiation $N_i$ from regular $S_N$-curves of monolithic metals [16].

The results of this first model verification presented in Fig. 12 illustrate that the increase of longitudinal FRP damage clearly influences the crack propagation in the metal layers. Due to the high load amplitude, a rather rapid shift from starting FRP damage (up to 18 % longitudinal degradation, Fig. 12 left screenshot) to final fiber failure (99 % longitudinal degradation, Fig. 12, right screenshot) can be observed. The offset of the two curves shown in Fig. 12 before the distinct kink of the red curve would be significantly larger in case of a slowly and continuously increasing FRP damage expected for lower load amplitudes in combination with a larger number of load reversals. With the onset of fiber failure the crack growth accelerates clearly (kink in the red curve, Fig. 12) while fiber failure spreads further. The slope of the blue curve (Fig. 12, no FRP damage) decreases continuously, indicating an increasing bridging efficiency for an increasing crack length.

This example supports clearly the general motivation of the proposed approach and shows the potential of the presented FML fatigue damage model. The results indicate, that FRP damage has the capability to influence significantly the crack growth processes in FMLs. Therefore, it seems mandatory to take into account possible FRP damage processes in order to predict fatigue damage for FMLs more accurately, see also [17].
In this work, a first version of a FE-based fatigue damage model for FMLs has been presented. A description of the methodical framework was given and first promising results were presented. The results indicate that FRP damage has the ability to influence the crack growth rates in FMLs significantly, which clearly motivates the consideration of possible FRP damage processes in terms of fatigue damage modeling for FMLs. However, the present framework represents a good basis for extensions that will be carried out in future work. The computational efficiency will be improved further, so numbers of load cycles relevant to many practical applications can be taken into account (HCF- / VHCF-regime). The implementation of an appropriate delamination approach is also of primary importance, since it directly affects the efficiency of the fiber bridging [16]. A third point of future improvement is the implementation of additional FRP damage modes to enable the analysis of additional load cases (e.g. shear), which are considered in the model of [5].

All these developments will expectably result in a FE-based fatigue damage model for FMLs that is able to predict any possible damage that can occur for cyclically loaded FMLs. The FE-framework allows the analysis of realistic structural components (e.g. bolted joints or cutouts) whereas the applicability of many established analytical damage models is generally limited to certain geometries (see for example [18]). The FE-framework – especially the detailed crack
modeling – also enables the analysis of more realistic loading scenarios including varying load amplitudes (see overload example in Section 3.1), which also feature crack closure phenomena. The presented results confirm the beneficial potential of FE-based approaches, as they have ability to account for complex fatigue damage analysis of realistic structural components.

REFERENCES


222
ENVIRONMENTAL EFFECTS ON HIGH TEMPERATURE FATIGUE OF CARBON-POLYIMIDE TEXTILE COMPOSITES FOR AIRCRAFT APPLICATIONS

F. Foti*, M. Gigliotti*, Y. Pannier*, D. Mellier*

* Physics and Mechanics of Materials Department Institut P’, CNRS, University of Poitiers, ISAE-ENSMA
1, Rue Clément Ader, Chasseneuil du Poitou, France
federico.foti@ensma.fr

Key words: Woven Organic Matrix Composite, Fatigue, μ-Computed Tomography, Digital Image Correlation.

Summary: This work aims at characterizing and modelling - for carbon fibre/organic matrix (polyimide) textile composites – the thermo-mechanical behaviour, the onset and the development of damage related to cyclic mechanical mechanisms (fatigue) under controlled (temperature and gas) environment.

1 INTRODUCTION

The use of fiber-reinforced composites in aircraft structural components has significantly increased in the past two decades [1] and future employment of these materials is foreseen for warm components (120°C < T < 300°C) as nacelles, fan cases, etc. During its life these components will be subjected to severe mechanical solicitations and to long-term superposition of mechanical cycling loading, thermo-oxidative ageing and thermal fatigue [2].

A consistent literature exists on fatigue of woven composites ([3]–[7]), on thermo-oxidative behaviour of polymers ([8]–[10]) and on organic matrix composites ([11], [12]), but the interaction between fatigue and thermo-oxidation remains poorly explored. For organic matrix based composites, the coupling between fatigue and thermo-oxidation of the resin, could lead to an acceleration of the fatigue degradation process.

This work aims at characterizing and modelling the thermo-mechanical behaviour, the onset and the development of damage related to cyclic mechanical mechanisms (fatigue) under controlled (temperature and gas) environment.

2 SPECIMENS AND TEST SETUP

The specimens are cut from 8 harness satin carbon fibre/polyimide matrix [0]₆ plates provided by Safran Nacelles. The polyimide resin has a glass transition temperature of 315°C and the composite has to be able to operate at temperatures up to 250°C. Off-axis [45]₆ specimens are cut from the plates in order to enhance the matrix oxidation effects on the macroscopic properties of the specimens.

Figure 1 illustrates the COMPTINN’ experimental setup, specifically developed in order to perform fatigue tests under controlled environment. An INSTRON 1251 hydraulic fatigue
Machine (F_max: 50 kN) is equipped with an aging chamber including a control of temperature and gas pressure (T_max: 350°C, p_max: 5 bar, environment: air, O_2, N_2, …). A system for gas supply and an electronic control heating is installed around the machine; this facility allows carrying out traction and fatigue test in controlled environment, achieving a maximal temperature of 350°C and a maximal gas pressure of 5 bar.

Since tests are performed at high partial gas pressure, the samples standing in a closed environmental chamber, non-contact measure methods must be employed. Full field digital image correlation (DIC) method is used to measure sample strain field and to characterize surface damage during the test: a high temperature resistant (max usage temperature: 650°C) speckle pattern sprayed on one surface of the specimens is employed. During the tests, a series of pictures with a pixel dimension of 6 µm/pixel, at a frequency of 6 seconds each, are taken using a D3X Nikon camera mounting a Sigma 150mm f/2.8 Macro zoom having a resolution of 6048x4032 pixel. Starting from these pictures, deformations in longitudinal and transversal directions are computed using CORRELA, a software dedicated to the digital image correlation developed at the PPRIME Institute ([13], [14]).

Figure 1: COMPTINN' experimental setup. Instron 1251 fatigue setup equipped by an environmental climatic chamber.

Figure 2: shows the RX Solution® X-Ray micro Computed Tomography available at the PPRIME Institute. μCT scans permits to perform a complete analysis of cracks distribution in the tested specimens. Scans (voxel dimension up to 8.78µm/voxel) are carried out using a voltage of 60kV and a current of 139µA.

Figure 2: RX Solution® X-Ray micro Computed Tomography facility – available at the Pprime Institute.
3 FATIGUE TESTS RESULTS

Tensile tests on composite specimens are performed at 250°C in order to fix mechanical parameters for fatigue tests. Figure 3 shows the tensile stress/strain curve for a [45] specimen.

In the first stage of traction the material behaviour is quite linear, then around 60-70 MPa it starts to become increasingly non-linear until 130 MPa (ultimate stress $\sigma_{ult}$) where the test is stopped. Fatigue tests are performed using a maximal load level $\sigma_f^{max}=0.6\sigma_{ult}$, a stress ratio $R=0.1$ and a frequency $f=2$ Hz. Fatigue tests are periodically cut off (around 20 times) to perform slow load/unload cycles for DIC measures; in addition, samples are periodically (4 times) removed from the test rig and subjected to ex-situ $\mu$CT scans. The interrupted fatigue test cycle is illustrated on the right side of Figure 3.

Fatigue tests were carried out at 250°C under three different environmental conditions: the 2 bar $O_2$ environment is strongly aggressive and should favour matrix degradation; the 2 bar $N_2$ is inert and may serve as a reference environment and under Air.

3.1 DIC results

DIC is used in this work to characterize surface damage of the specimens. A typical longitudinal strain field is illustrated in Figure 4. The surface cracks are more visible when the specimen is loaded with the maximum fatigue stress. For this reason, only the strain fields relative to maximum fatigue stress are studied in this paragraph.

DIC measures result in a strongly heterogeneous strain field, high strain zones are related to the architecture of the woven, DIC errors and cracks. To separate cracks from the two former sources of high strain zones, a quantification of the strains related to woven architecture and DIC errors is performed. For this analysis, an image of a strain field at the maximum fatigue load of a virgin specimen is reported in Figure 5. In the represented strain field the strain values fluctuations are related to DIC errors and woven features.

A statistical analysis allows quantifying this kind of fluctuations. The strain field of the specimen in Figure 5 has a mean value of 0.63 and a standard deviation of 0.18. The mean value is represented in Figure 5 by a red line. For the following analysis, this value for standard deviation will be added and subtracted from the mean value of the strain of each strain field relative to a load $\sigma=\sigma_f^{max}$. For the images relative to the same specimen the mean value will be re-calculated for each stop of the fatigue test, while the standard deviation, being dependent from correlation errors and weave pattern, will be used as a constant value. Crack strain values are generally higher than mean strain value of the image, hence only the strain values exceeding the described bounds and higher than the mean strain of the image
will be considered as cracks. This is equal to fix a threshold value on an image for a thresholding segmentation.

\[ \sigma = \sigma_{\text{MAX}}^f \]

Figure 4: DIC strain field at \( \sigma = \sigma_{\text{MAX}}^f \). Different sources of strain magnification: DIC algorithm errors, weave features and cracks in -45° direction.

Figure 5: DIC strain field on virgin specimen. Analysis and quantification of noise.

The second part of crack detection is based on image processing. A binary matrix representation of the remaining strain values is created and the image is analysed using Avizo9®. The binary image is segmented using simple thresholding as previously suggested. A Skeletonization is then carried out on the segmentation. By this operation, a line is traced inside each segmented zone, equidistant to shape boundaries of the element. At the end of this step, the DIC strain field is reduced in a map of segments.

Figure 6 shows the original strain field, the binarization (thresholding) and the skeletonization result. A MATLAB® script is finally employed to suppress segments.
oriented along directions different from the cracks directions (-45° in Figure 6), hence high strain zones oriented transversally will be not considered. The same script is finally employed to measure the length of the remaining segments, reported in blue in Figure 6 c).

![Image](image.png)

Figure 6: DIC image processing. a) Original strain field representation b) binarization result and c) skeletonization

The results of crack detection by DIC are shown in Figure 7. In terms of total crack length evolution, the damage for the three specimens is affected by the test environmental conditions.

The 2 bar O_{2} specimen fails after only 550k cycles, while the 2 bar N_{2} and the air specimens complete the test (1M cycles) without fail. The 2 bar O_{2} sample exhibits an evolution of the total crack length which is faster than the air specimens. The crack evolution for the 2 bar N_{2} specimen is quite slow compared to the 2 bar O_{2} and air specimens. Furthermore, after 800k cycles the 2 bar N_{2} specimen has a stable crack length evolution, while the air specimens show an acceleration of the damage accumulation.

![Image](image.png)

Figure 7: Crack length evolution measured by DIC. The damage evolution for the 2 bar O_{2} specimens is faster than the damage evolution encountered for the specimens tested in air and 2 bar N_{2}.

The DIC damage evolution is a surface measure. In the following paragraph a volume damage description is provided by μCT scans.
3.2 µCT results

In order to better identify the physical mechanisms (damage, degradation …) responsible for the behavior observed at the macroscopic sample scale, µCT scans are carried out on interrupted fatigue samples. These observations are useful to elucidate the chronology of damage/degradation onset and development and to detect the sites/scale at which such phenomena take place.

Figure 8 illustrates the crack segmentation process: in general, several types of damage occur during sample fatigue: cracks along the two fibers direction (in blue and green) appear inside all the plies, while cracks oriented in the direction perpendicular to the load direction (red), affect the resin rich regions only in the exterior plies.

The total damaged volume is then evaluated by applying the same process to the whole scanned sample volume.

![Figure 8: Segmentation process. On the left, segmented cracks in the ply and on the right segmented cracks with the transparent ply.](image)

Figure 9 shows the total damaged volume ratio ($D_{\mu CT}$) defined as the total damaged volume divided by the total scanned volume as a function of the number of cycles for the three tested specimens.

![Figure 9. Evolution of the damaged volume ratio as a function of the number of cycles.](image)

Also in terms of $D_{\mu CT}$ the environmental effects seem to affect damage evolution. For the 2 bar N$_2$ sample the evolution of $D_{\mu CT}$ shows a slight rise from the beginning until the end of the test. A faster evolution is then observed for the 2 bar O$_2$ sample, that fails at 550k cycles and finally, for the air sample, an evolution faster then, but close to, the 2 bar N$_2$ specimen is detected. The acceleration for damage evolution of the air specimen from 800k cycles does not appear as clearly as it is in Figure 7.

The indicator $D_{\mu CT}$ does not discriminate between different types of cracks. The damage variable $D_{\mu CT}$ gives a global quantification of the damaged volume for each specimen. A
more detailed description of the specimen damage is proposed in order to better understand how the environment affects damage evolution.

Figure 10 shows a 3D µCT reconstructed image of a damaged woven specimen. The zoom on the XZ and on the XY planes, reveals that the main damage form is intra-tow crack and the cracks are principally narrowed in the external tows. Further information issued from Figure 10 are related to cracks shape. Each crack has a regular path, that is, each crack is preferentially oriented in the belonging tow direction, and does not propagate through the specimen thickness.

![3D image reconstruction](image)

Figure 10: 3D image reconstruction. Two cut planes show typical intra-tow crack shape, location and dimensions: the cracks are oriented in the same direction of the belonging tow, furthermore cracks in surface tow are more opened than internal tow cracks.

The procedure adopted for crack segmentation allows separating the crack volumes for each tow direction and for each ply. The detailed damage quantification results from these separation, taking into account the through the thickness damage distribution. As illustrated in Figure 11, each ply of the woven is split in two semi-plies and for each semi-ply the tows are oriented along a preferential direction. In Figure 11 the woven plies are enumerated from one to six. The PLY1 and the PLY6 are the external plies, while the plies from two to five are internal plies. Moreover, in the PLY1, the semi-ply oriented along the -45° direction is the semi-ply directly exposed to the environment, while, for the PLY6, is the +45° semi-ply to be exposed to the environment.

![Woven architecture simplification](image)

Figure 11: Woven architecture simplification. Each ply is split in two semi-plies having the tows oriented in one preferential direction. For the tested specimens each ply is divided in a -45° semi-ply and a +45° semi-ply.
The simplification of dividing schematically the woven specimen in twelve semi-plies is used in Figure 11 to represent the damage volume evolution along the through the thickness direction of the specimen.

On the ordinate axis of the histograms in Figure 12, the damage for the single ply is calculated as:

\[ D_{\text{ply}}^{\mu\text{CT}} = \frac{V_{\text{CRACKS semi-Ply}}}{V_{\text{TOT Ply}}} \]  

where \( V_{\text{CRACKS semi-Ply}} \) is the crack volume in the semi-ply and \( V_{\text{TOT Ply}} \) is the volume of the whole ply. The damage distribution along the through the thickness direction resumed in the histograms of Figure 12 highlights that for each test condition, the damage volume is located in the tows directly exposed to the environment.

Figure 12 : Through the thickness crack distribution. For the three test condition the damage is concentrated on the external surfaces for all the test duration. This is not the case for 2 bar \( O_2 \) specimen at the end of the test, where an important damage volume is measured in the interior plies.

For the 2 bar \( N_2 \) and Air specimens the damage is mainly confined on the external semi-plies until the end of the test, while for the 2 bar \( O_2 \) specimen, the last \( \mu\text{CT} \) scan reveals that a consistent part of damage affects also the internal plies. Specimen surface observed by DIC is one of the two most damaged surfaces of the specimen, i.e. the \(-45^\circ\) tow direction belonging to the PLY1. The damage scenario in the 2 bar \( N_2 \) specimen is related to the woven architecture, while for the air and the 2 bar \( O_2 \) specimen is due to the woven architecture and to the environmental effects. The environment does not affect significantly the damage development in the external plies of the air specimen, while its effect is quite visible for the external and the internal plies of the 2 bar \( O_2 \) specimen. Apparently, the environment affect not only the damage volume, but also the damage distribution in the specimen.

### 3.3 DIC and \( \mu\text{CT} \) comparison

Figure 13 illustrate a segmentation result of a damaged specimen PLY1 in the \(-45^\circ\) direction. The zoom of the segmentation results show the cracks situated on the DIC observed zone of the same surface. Due to the different spatial resolution, longest cracks are
clearly visible using both µCT scan segmentation and DIC, while shortest crack are detectable only using the µCT segmentation procedure.

A comparison between the damage in the -45° tows of the PLY1 measured by µCT and by DIC is reported in Figure 14. Each graph has a double ordinate axis: in the former is reported the total crack length and in the second the PLY1 damaged volume calculated by the equation (1) for the PLY1. The scaling factor between the two ordinates is calculated considering that an entirely cracked tow, results in a $D_{\text{µCT,ply1}}$ between 0.08% and 0.12% (depending from the crack thickness).

The relatively most important results concern damage evolution: the µCT scans are limited in number during the fatigue test due to the complexity in remove/mounting the
specimens to perform the scan. DIC stops are more numerous, hence the data points obtained allow tracing a more detailed evolution for first ply damage during fatigue. The good agreement between DIC and µCT data illustrated in Figure 14 confirms that DIC is useful for surface damage characterization.

Graphs in Figure 14 compare two kinds of damage characterization that finally give similar information. The damage parameter $D_{\mu CT}$ gives a volumetric description of the damage, and finally is a volumetric ratio that should represent the damage state of the whole specimen. DIC gives a surface evolution/quantification of damage obtained studying a reduced zone of specimen if compared to µCT (see Figure 14).

A conclusion issued from this last observation is that the volumetric damage description through the $D_{\mu CT}$ is a good index of an average damage state of the specimen. For a very heterogeneous damage scenario as observed for the 2 bar N$_2$ specimen, the DIC results are quite far from the µCT ones, proving the importance to scan a large region of the specimen to have an appropriate average mean damage characterization.

The information provided by DIC on the PLY1 could be used to describe the damage development of the whole specimen keeping in mind that for a consistent part of fatigue tests the damage is narrowed on the external surfaces. Considering the volume of the whole specimen, the scaling factor will be 6 times the scaling factor used for comparison of the two measure methods employed for the graphs in Figure 14. Furthermore, to trace the graphs relative to the whole volume, the crack length measured by DIC will be multiplied by a factor 2, taking into account that the damage observed on the surface tow of the PLY1 and PLY6 has the same value; consequently the ordinate $L_{TOT}$ will be named $2L_{TOT}$. The resulting graphs are reported in Figure 15.

![Figure 15: Damage on the whole specimen volume quantified using DIC and µCT scan segmentation.](image)

The scaling factor and the assumption that the evolution of the damage for the PLY6 seem in good accord with the experimental segmentation results. For the graph relative to the 2 bar
F. Foti, M. Gigliotti, Y. Pannier, D. Mellier

N\textsubscript{2} the distance between the points obtained after 1M cycles could be justified as previously made by the heterogeneous distribution of the damage. The DIC results for the 2 bar O\textsubscript{2} specimen are validated by the \textmu{}CT results until 170k cycles. The scan performed after 550k cycles shows an important damage volume also for the internal plies, and no additional information on the damage development for these plies are available. This incertitude in damage evolution for the 2 bar O\textsubscript{2} specimen is represented in Figure 15 by a split of the dotted line describing the damage evolution. The factor 2 used to rescale DIC damage seems to include the PLY6 damage and the slow damage increasing of the internal plies. The damage on the PLY6 is ever around the 80\% of the damage in the PLY1, except for the Air specimen where damage development in PLY6 and PLY1 is quite similar. For this reason in the graph of Figure 15 related to Air specimen, DIC tends to overestimate damage development if compared to \textmu{}CT results.

The damage assessment performed by DIC for the tested material is a powerful method, it allows obtaining a more detailed damage evolution using a larger number of data point than the \textmu{}CT derived method. The limits of the explained method is clearly highlighted in the graph relative to the 2 bar O\textsubscript{2} specimen in Figure 15.

CONCLUSIONS

Fatigue tests in Air, oxidizing (2 bar O\textsubscript{2}) and inert (2 bar N\textsubscript{2}) environment are carried out to investigate thermo-oxidation/fatigue interaction. Failure of the 2 bar O\textsubscript{2} sample occurs after 550k cycles, while the 2 bar N\textsubscript{2} and Air samples have not failed after 1M cycles. DIC measures reveal a quite different damage evolution for the three tested specimens testifying higher degradation as a function of number of cycles for the 2 bar O\textsubscript{2} sample.

According to this observation, \textmu{}CT scans show a considerable rise of damage volume ratio in the interior plies for the 2 bar O\textsubscript{2} samples comparatively to Air and 2 bar N\textsubscript{2} samples, between 170k and 550k cycles. These observation allow concluding that environment affects not only damage evolution, but also damage distribution (damage morphology).

Moreover, DIC on specimen surface gives a damage evolution trend for the tested specimens that is in good agreement with the \textmu{}CT scan segmentation result for the most part of fatigue. For the 2 bar O\textsubscript{2} specimen at failure, a considerable part of damage in the interior plies results in a significant gap between damage detected by DIC and \textmu{}CT scan segmentation.

In future studies, a Finite Element model will be developed to better understand through comparative simulations the impact of a thermo-oxidizing environment on the observed behaviour.

Acknowledgments

Part of the present research has been carried out within the context of the FUI COMPTINN’ program, in collaboration with Airbus Group Innovation and SAFRAN Group. We acknowledge Aircelle, SAFRAN Group for providing the material under study. This work was partially funded by the EQUIPEX “GAP” (reference ANR-11-EQPX-0018) and LABEX “INTERACTIFS” (reference ANR-11-LABX-0017-01) French Governmental programs.
REFERENCES


MATERIAL PARAMETER IDENTIFICATION CHALLENGE AND PROCEDURE FOR INTRA-LAMINAR DAMAGE PREDICTION IN UNIDIRECTIONAL CFRP

Fabio Malgioglio∗,1, Delphine Carrella-Payan1, Benoît Magneville1, Laszlo Farkas1

1Siemens PLM Software
Leuven (Belgium), Toulouse (France)
*fabio.malgioglio@siemens.com

Keywords: Prepreg and Textile Composites, Progressive Failure Analysis, Model Identification.

Summary: Composite materials, due to their high specific stiffness and strength, are well suited for applications where the weight is a crucial design requirement. In particular, composites are nowadays becoming more important in fields where the reduction of energy consumption is desired, such as the automotive industry. Due to the intrinsic complexity of composites, the characterization of such materials implies a high number of tests and, as a consequence, a high cost of the design itself. Advanced simulation solutions are efficient and accurate support to reduce tests and cost when combined with an experimental campaign. CDM models based on Ladevèze [1] for unidirectional materials and on Hochard [2] for woven fabrics, available in Siemens PLM Software (LMS Samtech Samcef), were proven to efficiently reproduce the complex behavior of carbon fiber laminated composites under tensile or compressive loading [3]. In addition, the Parameter Identification procedure (PI) for these models is based on few coupon tests on standard specimens with three different stacking sequences, which makes the models industrially applicable. Nevertheless, the interpretation of the experimental data may lead to a broad scatter in the identified parameters, which makes the PI process not straightforward. The purpose of this paper is to present and illustrate a Parameter Identification process for the determination of the intra-laminar properties of carbon fiber unidirectional composites. Different data reduction methods are herein critically discussed. In addition, sensitivity calculations are performed in order to assess the effect of uncertainties (e.g. originating from the experimental measurements) on the material parameters.

1. INTRODUCTION

Advanced composites, due to their specific stiffness and strength, are nowadays common materials in special applications where material performances and weight are crucial design requirements, for example in racing cars and aircrafts. The automotive industry is more and more interested in composite materials to reduce drastically the fuel consumption and emissions, also in view of the 2°C limit of global warming set in the Paris Climate Conference. The simulation of the complex material behavior is necessary in the design stage of composite components in
order to reduce the number of tests. Several numerical models have been developed for the prediction of failure of composite materials [1, 2, 4, 5]. Among them, Ladeveze et al. [1] proposed a model based on continuum damage mechanics able to reproduce the behavior and failure of unidirectional materials, including damage and plasticity. When choosing a material model rather than another one, three aspects should be taken into account. The first one is the accuracy of the model, i.e., the capability of reproducing the physics. The second aspect is the computational efficiency. The last one is the input data identification for the model. While accuracy and efficiency are accounted for in the model assessment, the input data identification procedure, or parameters identification, is considered in few papers [6]. The simulation methodology proposed in Ladeveze’s model and in Hochard’s model [2], available in Siemens PLM Software, were proven to provide both accuracy and numerical efficiency [3, 7]. At the same time, these methods are industrially applicable due to the limited number of material parameters and to the low number of tests required for their full identification. Nevertheless, the parameter identification (PI) presents several challenges, since parameters are affected by uncertainty. Uncertainty can be due to the test data variability, as well as errors in measurements during testing. In addition, the choice of data reduction methods can be, to some extent, left to the discretion of who performs the PI, leading to the identification of different material parameters for the same material system. Furthermore, the accumulation of uncertainties in the PI can affect the accuracy of the simulations and lead to conservative or even non-conservative results.

In this paper, the full parameter identification process from tests results and the most suitable data reduction methods are discussed for the intra-laminar behavior of unidirectional carbon fiber materials. Also, a sensitivity study on the parameters is performed in order to assess the effect of uncertainties related to measurements on the PI and to provide consistency to the calibration process.

2. INTRA-LAMINAR DAMAGE MODEL

The damage model, based on continuum damage mechanics (CDM), was first introduced by Ladeveze [1]. The model is able to account for damage and plasticity in a UD ply in longitudinal, transverse and shear directions. The elastic strain potential $e_d$ describes the state of the material in a certain loading condition and damage level. In two dimensions it is defined as:

$$e_d = \frac{\sigma_{11}^2}{2(1 - d_{11})E_1^0} - \frac{\nu_1}{E_1^0} \alpha_{11} \sigma_{22} - \frac{\langle \sigma_{22} \rangle^2_+}{2(1 - d_{22})E_2^0} + \frac{\langle \sigma_{22} \rangle^2_-}{2E_2^0} + \frac{\sigma_{12}^2}{2(1 - d_{12})G_{12}^0}$$

(1)

where $E_1^0$, $E_2^0$ and $G_{12}^0$ are the elastic moduli of the undamaged material, $d_{11}$, $d_{22}$ and $d_{12}$ are the damage variables in the longitudinal (1), transverse (2) and shear (12) directions respectively. The quantity $\langle X \rangle_+$ (or $\langle X \rangle_-$) is equal to $X$ if $X$ is positive (or negative) and 0 otherwise. Thermodynamic forces $Y_i$ are defined as the derivatives of $e_d$ respect to $d_i$:

$$Y_i = \frac{\sigma_i^2}{2(1 - d_i)E_i^0}$$

(2)
Table 1. Parameters for the intra-laminar behavior of the UD composite in Samcef

\[ R(p) = K p^m + R_0 \]  
where \( R(p) \) is the yield function, \( K, m \) and \( R_0 \) are material parameters and \( p \) is the cumulative plastic strain. Considering pure shear loading, \( p \) is defined as:

\[ p = \int (1 - d_{12}) \gamma_{12}^p \]  

being \( \gamma_{12}^p \) the plastic strain is shear. The yielding condition is met when

\[ \tilde{\sigma}_{12} = R(p) \]  
where \( \tilde{\sigma}_{12} \) is the effective shear stress and takes into account damage as follows:

\[ \tilde{\sigma}_{12} = \frac{\sigma_{12}}{1 - d_{12}} \]
The effect of plasticity is taken into account also in other loading conditions through coupling coefficients, which will be discussed in more detail in Section 4.5. A complete list of the needed parameters is presented in Table 1.

3. EXPERIMENTAL SETUP AND GUIDELINES

Composite coupons are tested in tension and in compression. Tensile tests are performed according to the ASTM D3039 [8] and D3518 [9], while compressive tests on ASTM D6641 [10]. All the specimens should be equipped with strain gauges aligned in the longitudinal direction and transverse direction (for tension only) on both sides in order to measure $\varepsilon_L$ and $\varepsilon_T$. Alternatively, strains can be measured by Digital Image Correlation (DIC). For each specimen, the cross section $A$ is measured before testing. The longitudinal stress $\sigma_L$ is calculated as $P/A$, where $P$ is the load applied by the testing machine.

In tension, three different kinds of specimens are tested: $[\pm 45]_{ns}$, $[0/90]_{ns}$ and $[\pm 67.5]_{ns}$ in order to capture the different material behavior in the main in-plane directions, i.e. longitudinal tension, transverse tension and in-plane shear. The tests in compression are performed on $[0/90]_{ns}$ laminates only. The tests are:

- Monotonic tensile test on $[\pm 45]_{ns}$ specimens
- Monotonic tensile test on $[0/90]_{ns}$ specimens
- Monotonic tensile test on $[\pm 67.5]_{ns}$ specimens
- Cyclic tensile test on $[\pm 45]_{ns}$ specimens
- Cyclic tensile test on $[\pm 67.5]_{ns}$ specimens
- Combined Load Compression test on $[0/90]_{ns}$ specimens

The levels of load for each cycle must be decided on top of the results of monotonic tensile tests in order to have a proper set of data points for damage evaluation. It is advised to set at least 5 cycles, some of which should have maximum loads close to the maximum load measured in monotonic tests. In the case of $[\pm 45]_{ns}$ specimens, the maximum load is defined in the standard as the minimum between the absolute maximum load and the load registered at 5% strain.

4. PARAMETER IDENTIFICATION PROCEDURE

The parameter identification (PI) is the process in which all the material properties needed to feed the model are determined. All the properties for the damage model are described in Table 1. Ladeveze et al [1] proposed to limit the number of specimens required by using three different stacking sequences: $[0/90]_{ns}$, $[\pm 45]_{ns}$ and $[\pm 67.5]_{ns}$. All the material parameters, listed in Table 1, are determined by means of tensile and compressive tests, either monotonic or cyclic, described in Section 3. The different parameters can be divided in different groups: elastic properties, damage parameters, fiber non-linearity coefficients, plasticity parameters and coupling coefficients.
4.1 Identification of Elastic Properties

The elastic behavior of an orthotropic lamina can be described by the Hooke’s law:

\[
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\tau_{12}
\end{bmatrix} =
\begin{bmatrix}
Q_{11} & Q_{12} & 0 \\
Q_{21} & Q_{22} & 0 \\
0 & 0 & Q_{66}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_1 \\
\varepsilon_2 \\
\gamma_{12}
\end{bmatrix}
\]  
(7)

where \(\sigma_1\) and \(\sigma_2\) are the in-plane stresses in the longitudinal and transverse direction respect to the fiber, \(\tau_{12}\) is the in-plane shear, \(\varepsilon_1\), \(\varepsilon_2\) and \(\gamma_{12}\) are the in-plane strains in longitudinal, transverse, and shear direction and \(Q\) is the stiffness matrix. By identifying the stiffness matrix, all the properties of the lamina are also identified. The shear modulus of the undamaged material \(G_{12}^0\), equal to \(Q_{66}\), is directly identified with tensile tests on \([\pm 45]_{ns}\) specimens. Typically, the stress-strain curve for this configuration is highly non-linear due to damage and plasticity taking place in the composite, therefore, \(G_{12}^0\) must be calculated at the beginning of the curve.

The calculation can be carried out in two different fashions: the chord method and the tangent method. With the first method, based on the ASTM standard, the chord modulus \(G_{12}^{chord}\) is defined as:

\[
G_{12}^{chord} = \frac{\Delta \tau_{12}}{\Delta \gamma_{12}}
\]  
(8)

where \(\tau_{12}\) is the engineering shear stress, defined as half of \(\sigma_L\), and \(\gamma_{12}\) is the engineering strain, defined as \(\varepsilon_L - \varepsilon_T\). In (8) the considered shear strain interval \(\Delta \gamma_{12}\) is 0.4 \(\pm\) 0.02\% long, starting from a strain level between 0.15 and 0.25\%. Alternatively, the shear modulus \(G_{12}^{tang}\) can be calculated with the tangent method as the derivative of the polynomial fit of order 2 of the stress-strain curve at the origin (Figure 1).

Figure 1. Graphic representation of the chord and tangent moduli
With the test on \([0/90]_{ns}\) the quantities \(Q_{11} + Q_{22}\) and \(Q_{12}\) can be identified, similarly to \(G^{0}_{12}\), with the chord and tangent methods. With the chord method we have:

\[
\begin{align*}
[Q_{11} + Q_{22}]_{chord} &= \frac{2\Delta \sigma_L \Delta \varepsilon_L}{\Delta \varepsilon_L^2 - \Delta \varepsilon_T^2} \quad (9) \\
[Q_{12}]_{chord} &= \frac{-\Delta \sigma_L \Delta \varepsilon_T}{\Delta \varepsilon_L^2 - \Delta \varepsilon_T^2} \quad (10)
\end{align*}
\]

where \(\Delta \sigma_L\) and \(\Delta \varepsilon_T\) correspond to a range \(\Delta \varepsilon_L\) between 0.1% and 0.3%, as recommended in the ASTM D3039 standard. With the tangent method, (9) and (10) become:

\[
\begin{align*}
[Q_{11} + Q_{22}]_{tang} &= \frac{2\sigma_L \varepsilon_L}{\varepsilon_L^2 - \varepsilon_T^2} \quad (11) \\
[Q_{12}]_{tang} &= \frac{-\sigma_L \varepsilon_T}{\varepsilon_L^2 - \varepsilon_T^2} \quad (12)
\end{align*}
\]

In (11) and (12), \(\varepsilon_L\) and \(\varepsilon_T\) are the longitudinal and transverse strains corresponding to a unitary stress \(\sigma_L\). The information obtained from the tensile test on \([0/90]_{ns}\), i.e. \(Q_{11} + Q_{22}\) and \(Q_{12}\) together with those from tensile tests on \([\pm 67.5]_{ns}\), are used to identify \(Q_{11}\) and \(Q_{22}\) and therefore the Elastic Properties. \(Q_{11}\) is calculated with the two methods. With the tangent method we obtain:

\[
\begin{align*}
[Q_{11}]_{tang} &= \frac{\sigma_L \varepsilon_L - \left[ s^4(Q_{11} + Q_{22}) + 2c^2s^2Q_{12} + 4c^2s^2Q_{66}\right] \varepsilon_L^2}{\left[ c^4 - s^4\right] \left[ \varepsilon_L^2 + \varepsilon_T^2\right]} - \frac{\left[ c^4(Q_{11} + Q_{22}) + 2c^2s^2Q_{12} + 4c^2s^2Q_{66}\right] \varepsilon_T^2}{\left[ c^4 - s^4\right] \left[ \varepsilon_L^2 + \varepsilon_T^2\right]} \quad (13)
\end{align*}
\]

where \(s = \sin(\theta)\) and \(c = \cos(\theta)\), being \(\theta\) the fiber orientation angle. To obtain \(Q_{11}\) with the chord method, \(\Delta \sigma_L\), \(\Delta \varepsilon_L\) and \(\Delta \varepsilon_T\) must be used in equation (13) instead of \(\sigma_L, \varepsilon_L\) and \(\varepsilon_T\). \(\Delta \varepsilon_L\) should be from 0.1% to 0.3%, as recommended in the standard.

\(Q_{22}\) is calculated simply by difference between (11) and (13). It must be noted that, being related to the stiffness in the longitudinal and transverse direction respectively, \(Q_{11}\) is one order of magnitude greater than \(Q_{22}\). The quantity measured in tests (i.e. \(Q_{11} + Q_{22}\)) is subjected to measurement errors, which may be of the same order of magnitude of \(Q_{22}\). The indirect calculation of \(Q_{22}\) is therefore critical and a calibration of this value with FE correlation is advised.

Once the stiffness matrix is identified, the compliance matrix of the ply is calculated by inversion:
\[ Q^{-1} = S = \begin{pmatrix} \frac{1}{E_{11}} & -\frac{\nu_{12}}{E_{11}} & 0 \\ -\frac{\nu_{12}}{E_{12}} & \frac{1}{E_{22}} & 0 \\ 0 & 0 & \frac{1}{Q_{66}} \end{pmatrix} \] (14)

From (14) the calculation of the engineering constants for the orthotropic lamina is straightforward.

In this section, two methods for the calculation of the elastic properties were shown. The difference between the two methods is more evident in the case of shear test. As shown in Figure 1, \( G_{12}^{\text{chord}} \) is smaller than \( G_{12}^{\text{tang}} \) and can better represent the material behavior for moderately small strains (e.g. \(< 0.6\%\)). However, the choice of the tangent method over the chord method presents several advantages. Firstly, the material may exhibit non-linearities (i.e. damage and plasticity) at very low strains, which can be simulated only if \( G_{12}^{\text{tang}} \) and its degradation are considered. Secondly, \( G_{12}^{\text{chord}} \) is dependent on the choice of the strain interval, while \( G_{12}^{\text{tang}} \) is not. With the tangent method, also the first loading cycle of cyclic tensile tests (Section 3) can be used for the determination of the elastic properties even if the applied strain is low. In this way, the shear modulus is calculated over a greater number of specimens. An additional advantage of \( G_{12}^{\text{tang}} \) becomes clear when calculating the shear damage coefficients (Section 4.2). The use of \( G_{12}^{\text{chord}} \) instead of \( G_{12}^{\text{tang}} \) requires additional care in the PI. In fact, if the modulus calculated at the first loading cycle (Section 4.2) is greater than \( G_{12}^{0} \), the corresponding damage coefficients will be negative. In conclusion, the tangent method is suitable for the damage model, while the chord method is more adequate for linear analyses and for designing the laminate stiffness, as well as for comparison to data of different materials tested according to ASTM standard.

4.2 Identification of Damage Parameters

The values of thermodynamic forces (Section 2) correspondent to damage initiation and complete failure in the different material directions must be identified. Let us consider the material loaded in tension in the fibers direction. Carbon/epoxy UD materials are considered to be brittle, so damage evolution can be neglected and \( d_{11} \) is equal to 1 when it fails and 0 otherwise. To obtain the tensile damage properties, tests on \([0/90]_{ns}\) laminates are used. The maximum load in a single ply \( \sigma_{11}^{\text{max}} \) is calculated by Classic Lamination Theory (CLT) and used in (2) to obtain \( Y_{11}^{t} \). In compression, \( Y_{11}^{c} \) is calculated by data from the CLC tests. The stress \( \sigma_{11}^{c} \) in the plies at 0\(^{\circ}\) is calculated with CLT using the elastic properties defined in Section 4.1.

The stress-strain curves of cyclic tests on \([\pm 45]_{ns}\) specimens are needed to identify damage properties in shear. For every cycle, the intersections of the unloading curve and the loading of the next cycle must be found (Figure 2). In this way, the reduced stiffness \( G_{12} \), the damage variable \( d_{12} \) and the correspondent thermodynamic force \( Y_{12} \) can be calculated in every specimen. By plotting \( d_{12} \) with respect to \( \sqrt{Y_{12}} \) (Figure 2), the damage evolution law can be extrapolated. \( Y_{12}^{0} \) is the intersection of the damage evolution curve with the horizontal axis. A polynomial
fit of order 2 can be used to describe the damage behavior in shear. Failure takes place when $Y_{12}$ reaches the critical value of $Y_{12}^c$, which can be defined as the maximum value of $Y_{12}$ from the plot in Figure ??.

The material loaded in shear experiences changes in fiber angle due to fiber movements inside the matrix in case of ductile matrix, weak fiber-matrix interfaces or thick laminates [9]. If this happens, the assumption of pure shear is no longer valid. For this reason the standard recommends that the maximum strain measured cannot exceed 5% (see Section 3) even if the specimen can resist higher loads. This must be taken into account when identifying the damage parameters in shear: the real value of $d_{12}$ at failure cannot be calculated, therefore, $Y_{12}^s$ calculated with (2) is conservative. As an alternative, $Y_{12}^s$ can be calculated with using the absolute maximum shear stress measured in the monotonic shear test (see Section 3) and the absolute maximum damage variable $d_{12}$ measured. To have a more accurate estimation of $Y_{12}^s$, a calibration with FE correlation is advised. The damage properties in the transverse direction can be extracted from test data on $[±67.5]_ns$ coupons. Here, from $\sigma_L$, $\varepsilon_L$ and $\varepsilon_T$, $\sigma_{12}$, $\sigma_{22}$, $\varepsilon_{12}$ and $\varepsilon_{22}$ can be obtained via CLT. The damage threshold value $Y_{22}^s$ is found similarly to $Y_{12}^s$ with equation (2).

4.3 Non-linearity in the fiber direction

In experiments, a small variation of stiffness in the fiber direction (an increase in tension and decrease in compression) is observed. These variations can be approximated linearly proportional to the longitudinal stress in the ply (Figure 4). $\xi^+$ and $\xi^-$ are obtained from the plot of the modulus $E_1$ respect to $\sigma_{11}$:

$$E_1 = E_1^0 + \langle \sigma_{11} \rangle + \xi^+ + \langle \sigma_{11} \rangle - \xi^-$$

where $E_1$ is calculated as the ratio between $\sigma_{11}$ and $\varepsilon_L$. 

\[ \text{Figure 2. Stiffness reduction from a cyclic test on a } [±45]_ns \text{ coupon} \]

\[ \text{Figure 3. Damage evolution in Shear} \]
4.4 Identification of Plasticity Parameters

For the identification of the plasticity parameters in shear, the cyclic shear test is exploited. At every cycle, the damage variable \( d_{12} \) and the plastic strain \( \gamma_{p12} \) can be calculated from the plot in Figure 2. The cumulative plastic strain, defined in eq (4), is obtained by integrating \( d_{12} \) with respect to \( \gamma_{p12} \). The function to be integrated is obtained through a polynomial fit of order 2 of the experimental data. The effective stress, which takes into account damage, is defined in (6). The effective stress \( \tilde{\sigma}_{12} \) and cumulative plastic strain \( p \) data points for every cycle in every specimens must be plotted in a Plasticity Master Curve (Figure 5).

Plasticity parameters for the plasticity law defined in (3) are identified with the power fit of the Plasticity Master Curve. The value \( R_0 \) is the intersection of the curve with the vertical axis and physically represents the shear yielding. The plasticity master curve is very steep at the beginning, which makes the identification of \( R_0 \) strongly dependent on the data points. The direct application of (3) may lead to negative values of \( R_0 \), which is clearly non-physical. In this case, (3) can be modified as follows:

\[
R = Kp^m
\]

and a small arbitrary positive value of \( R_0 \), e.g. smaller than 1MPa should be chosen (for numerical reasons).

4.5 Identification of Coupling Coefficients

The coupling between thermodynamic forces is defined as:

\[
Y = Y_{12} + b_2 Y_{22}
\]
For a laminate loaded in pure shear $Y_{22}$ vanishes and $Y = Y_{12}$. For different stacking sequences, $b_2$ must be identified. The cyclic tests on $[\pm 45]_{ns}$ and $[\pm 67.5]_{ns}$ are used to identify the coupling between the thermodynamic forces. $d_{12}$ is plotted with respect to $Y$ for the tests on $[\pm 4.5]_{ns}$ and $[\pm 67.5]_{ns}$. $b_2$ is changed until the error between the two curves is minimized (Figure 6).

![Figure 6. Determination of $b_2$ parameter](image)

The parameter $b_3$ is the linear coupling between $d_{12}$ and $d_{22}$:

$$d_{22} = b_3 d_{12}$$  \hspace{1cm} (18)

$d_{12}$ and $d_{22}$ can be found from tests on $[\pm 67.5]_{ns}$, while $b_3$ is simply the slope of the plot $d_{12}$-$d_{22}$ (Figure 7).

The coupling coefficient for plasticity $a$ must be found by FE calibration by simulating monotonic tensile tests on $[\pm 67.5]_{ns}$ specimens.

5. SENSITIVITY STUDY OF MATERIAL PARAMETERS

The PI described in Section 4 provides the material properties needed for the simulation of the ply behavior. Nevertheless, some parameters, e.g. $E_2$, are subjected to uncertainty due to the variability of experimental tests measurements and can be adjusted to obtain a better curve fitting. Such modifications, however, affect the identification of all the parameters depending on the changed value.

It is therefore useful to assess the parameters sensitivity due to uncertainty of measurements in the experimental tests. The effect of the variation of directly measured properties (i.e. the material stiffnesses and the coupons strengths) on the derived parameters is studied and presented in Table 2. The measured properties are changed one by one of 5% and for each variation a new PI is performed. The most affected parameters are $Y_{11}^t$, $Y_{11}^c$, $Y_{22}^s$ and $Y_{12}^s$. 

![Figure 7. Determination of $b_3$ parameter](image)
### Table 2. Table of error of the sensitivity study

<table>
<thead>
<tr>
<th>Property</th>
<th>Error %</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_1^0$</td>
<td>5.00</td>
</tr>
<tr>
<td>$E_2^0$</td>
<td>- 5.00</td>
</tr>
<tr>
<td>$G_{12}^0$</td>
<td>- 5.00</td>
</tr>
<tr>
<td>$\sigma_{L,[0/90]}^{t,\max}$</td>
<td>- 5.00</td>
</tr>
<tr>
<td>$\sigma_{L,[\pm45]}^{\max}$</td>
<td>- 5.00</td>
</tr>
<tr>
<td>$\sigma_{L,[\pm67.5]}^{\max}$</td>
<td>- 5.00</td>
</tr>
<tr>
<td>$\nu_{12}$</td>
<td>5.00</td>
</tr>
<tr>
<td>$Y_{11}^t$</td>
<td>-0.77</td>
</tr>
<tr>
<td>$Y_{11}^{c}$</td>
<td>0.05</td>
</tr>
<tr>
<td>$Y_{22}^0$</td>
<td>-4.76</td>
</tr>
<tr>
<td>$Y_{12}^t$</td>
<td>-4.76</td>
</tr>
<tr>
<td>$Y_{12}^{c}$</td>
<td>-4.76</td>
</tr>
<tr>
<td>$K$</td>
<td></td>
</tr>
<tr>
<td>$m$</td>
<td></td>
</tr>
<tr>
<td>$R_0$</td>
<td></td>
</tr>
<tr>
<td>$b_2$</td>
<td>4.76</td>
</tr>
<tr>
<td>$b_3$</td>
<td>-</td>
</tr>
</tbody>
</table>

6. CONCLUSIONS

In the present paper, the parameter identification procedure for the intra-laminar behavior of unidirectional carbon fiber composites according to Ladeveze’s model is presented. Two data reduction methodologies, the chord method and the tangent method, were discussed and the latter was proven to be suitable for the PI of the damage model. The parameters $E_2$, $Y_{22}^s$ and $Y_{12}^s$ were proven sensitive in this PI approach due to their mathematical derivation, leading to conservative values of $Y_{22}^s$ and $Y_{12}^s$.

The uncertainties due to the mathematical formulation and to the measurements from tests can affect the PI and, when necessary, a better set of material properties can be obtained by calibration with FE correlation on tests. However, the calibration process should consider the effect of the modification of each parameter on the other properties. For this reason a sensitivity study on the material parameters was herein proposed. It is shown how a variation of the measured parameters, i.e. the material stiffnesses and strengths of the tested laminates, affect all the derived properties. In particular, 5% errors on the laminates strengths have up to 10% effect on the threshold values of the thermodynamic forces $Y_i^s$.

The variability of the input parameters of the model discussed in this paper and its effect on FE simulations will be investigated in future studies.
References


DESIGN AND OPTIMISATION OF A NOVEL INTERLOCKING JOINING TECHNOLOGY FOR COMPOSITE-METAL STRUCTURES

Michael C. Corbett*, Conor T. McCarthy*

*Bernal Institute, School of Engineering, University of Limerick, Limerick, Ireland
michael.corbett@ul.ie

Keywords: Continuum damage model, Cohesive zone model, Hybrid joints

Summary: A novel concept is presented for joining composites to metals in lightweight, hybrid structures. The concept employs interlocking morphology formed on the surfaces of composite (female) and metallic (male) adherends that are coupled with a layer of adhesive so that they mechanically interlock in shear. Miniature, single-lap adhesive joint specimens are employed with a single truncated square pyramid interlocking profile, centred in the bond area. Finite element (FE) models of the interlocking joint architecture are developed in order to assess the mechanical performance of the concept. The modelling approach incorporates an intralamellar continuum damage model to account for damage in the composite material, a cohesive zone damage model to represent damage and fracture propagating through the adhesive, and elastic-plastic behaviour to simulate the mechanical response of the metal. Herein, the accuracy of the model is validated against experimental results, showing strong correlation in terms of stiffness, strength and damage predictions. The concept demonstrates improvements of up to 18.3% in terms of the joint’s ultimate failure load when compared to a standard adhesively bonded joint. Importantly, the concept also shows the potential for a shift in failure mode, from cohesive failure, to composite net-tension or shear-out type failure mode; similar to what may be observed for mechanically fastened joints. This indicates that the failure mode of the joint can be controlled through the design of the interlocking morphology; which provides significant improvements in the joint’s durability for optimal interlocking designs.

1 INTRODUCTION

Joining represents one of the greatest challenges in the design of hybrid, lightweight, composite - metal structures, as the strength of a joint dictates the strength and efficiency of the surrounding structure. In addition, joining provides numerous advantages; including: the ability to create complex parts and to join dissimilar materials, it facilitates disassembly, and importantly in composite structures, it arrests the propagation of fracture and provides structural impact and damage tolerance beyond that inherent to the materials of construction. The current industry standard for structural joining depends on mechanical fastening and adhesive bonding. These techniques have undergone substantial development in the past number of decades and hence, are approaching the upper limit of their potential performance. Despite this, each technology still presents inherent weaknesses. Mechanical fastening is inefficient; adds weight and introduces stress concentrations to the structure. Adhesive bonding provides improved efficiency but
presents difficulty in achieving consistent strength, which adversely affects reliability and leads to the over-design of structures. Recently, a number of hybrid joining processes have emerged which combine the positive traits of the fundamental joining techniques; such as weld-bonded and bonded-bolted joints. In the present work, a novel, hybrid mechanical-adhesive joining concept is explored. The concept is based on the fact that the central overlap region of an adhesively bonded joint is relatively inactive for load transfer. In order to activate this area, interlocking profiles are formed on the surfaces of male (metallic) and female (composite) adherends and coupled with a layer of adhesive. Thus, the hybrid joint provides the mechanical interlocking and added reliability of mechanical fastening together with the efficiency of bonding.

The present authors have previously investigated the concept for an interlocking joint with metallic male and female adherends; and subsequently optimised the design of the interlocking morphology [1]. The results demonstrated impressive improvements in both the joint’s ultimate failure load and the work required for fracture. Herein, the concept is investigated through finite element analysis of the joint architecture, in a hybrid composite-metal joint configuration. Sophisticated material models are employed in order to simulate deformation and damage of the composite, adhesive and metal; making it possible to accurately capture the mechanical response of the joint until catastrophic failure. Thus, it is possible to explore the role of the interlocking geometry on stress distribution within the adhesive, crack propagation through the adhesive, and the progression of damage in the composite material. This paper presents the validation of the proposed FE model against results from an associated experimental investigation, with the aspiration that the FE model may be subsequently used to optimise the hybrid joining concept numerically.

2 PROBLEM DESCRIPTION

The subject of the present investigation is a hybrid, composite-metal, miniature adhesively bonded single-lap joint (SLJ), as shown in Figure 1a. This specimen geometry was chosen corresponding to an analogous experimental investigation, as it allows for in-situ testing within an SEM chamber [2]; permitting real-time observation of damage progression (i.e. adhesive fracture, matrix cracking, and delamination) occurring at the micro-scale. The joint’s interlocking morphology is represented in its most fundamental form, as a single truncated square pyramid profile, centred in the 35mm² joint overlap area. This allows for effective investigation of the performance of the concept and the deformation and damage mechanisms occurring around a single profile. The design of this geometry may be described by four factors; length ($X_1$), depth ($X_2$), width ($X_3$), and inclination angle ($X_4$), which are based on the female adherend, as shown in Figure 1b.

The female adherend is characterised by a depression in its surface corresponding to the interlocking morphology and was manufactured from a carbon fibre/epoxy matrix system, HTA/6376. Each lamina has a nominal thickness of 0.125 mm. Thus, the resulting laminate had 16 plies, stacked in a quasi-isotropic configuration, [0/0/90/90]$_{2s}$. The depression in the surface of the experimental female adherend was manufactured through a laser machining process similar to that adopted by Leone et al. [3]. The male adherend is distinguished by a protruding
interlocking profile which is defined to fit the female adherend, with a constant clearance of 55 µm. It is machined from an aluminium alloy, AA5754 [4]. Both adherends are coupled with a constant thickness adhesive layer, which fills the 55 µm [2] clearance space between the interlocking surfaces (see Figure 1a). The adhesive system employed is a bi-component structural epoxy resin, Loctite® Hysol 9466 [5].

![Figure 1: Miniature interlocking SLJ (a) geometry and boundary conditions (dimensions in millimetres) and (b) truncated, square pyramid interlocking profile geometry and corresponding geometric factors.](image)

There were four miniature adhesively bonded joint configurations tested during the associated experimental investigation, three interlocking examples and a corresponding, baseline (C0), standard adhesively bonded joint. The geometry of the interlocking joints is described per Table 1, comprising of a rectangular interlocking profile, which is longer in the loading direction (C1), a profile which is longer transverse to the loading direction (C2), and a smaller, square profile (C3). These designs were set out to compare the performance of a diverse set of interlocking designs. The specimens are compared to the baseline in each case in order to distinguish improvement in performance for the interlocking adhesive joint concept.

<table>
<thead>
<tr>
<th>Joint</th>
<th>(X_1) [mm]</th>
<th>(X_2) [mm]</th>
<th>(X_3) [mm]</th>
<th>(X_4) [°]</th>
</tr>
</thead>
<tbody>
<tr>
<td>C0*</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>C1</td>
<td>3.0</td>
<td>0.75(\dagger)</td>
<td>1.2</td>
<td>85.0</td>
</tr>
<tr>
<td>C2</td>
<td>1.2</td>
<td>0.75(\dagger)</td>
<td>5.5</td>
<td>85.0</td>
</tr>
<tr>
<td>C3</td>
<td>1.5</td>
<td>0.75(\dagger)</td>
<td>1.5</td>
<td>85.0</td>
</tr>
</tbody>
</table>

* Standard miniature adhesive joint.
\(\dagger\) Equivalent to 6 ply thicknesses.

3 FINITE ELEMENT MODEL

A miniature adhesive joint model was developed in the commercial FE software, Abaqus® [6]. The model was composed of separate parts for the male and female adherends. The adhesive was represented by a solid orphan mesh, offset from the bond surface female adherend.
During analysis, the parts were tied together with surface based tie constraints [6]. The parts were idealised in order to facilitate meshing; machining fillets, which were present in the experimental specimens, were excluded and the joint was modelled with no adhesive fillet. Model creation and meshing were automated through a Python™ script. Global and local nodal seeds were appropriately assigned such that mesh density was increased at the ends of the joint overlap, where large gradients in strain localise, and in the vicinity of the interlocking profile, to accurately capture deformation and damage of the composite material.

The response of the metallic, male adherend was characterised by elastic-plastic behaviour, incorporating von Mises yield criteria and isotropic strain hardening, which was defined through Holloman’s equation, per Eq. 1.

\[
\sigma = K\varepsilon_p^n
\]

where \(\sigma\) is the true stress, \(K\) is the strength coefficient, \(\varepsilon_p\) is the true plastic strain and \(n\) is the strain hardening exponent.

The stress-strain response of AA5754 was produced from experiments; salient mechanical properties are summarised in Table 2. This material was represented by 8-node, linear, brick elements with reduced integration and hourglass control (C3D8R) [6]. No damage model was implemented in the metal as it was assumed that damage would develop in the composite adherend or cohesive failure would occur prior to failure of the aluminium.

The composite, female adherend was represented by a layered model, incorporating each lamina as an orthotropic material, thus accounting for the variation in stiffness through the thickness of the adherend. Each lamina was represented by a single layer of 8-node, linear, brick elements with reduced integration and enhanced hourglass control (C3D8R) [6]. The mechanical response of this material was represented by an intra-laminar continuum damage model (CDM). A physically based damage model was employed. At the ply level, damage is assumed to take the form of matrix micro-cracking, fibre-matrix de-bonding and fibre fracture. The model includes the effect of shear-transverse behaviour coupling and plasticity, to account for irreversible strains in undamaged parts of the matrix, and incorporates non-linear in-plane shear behaviour. The Hashin criteria [7] is employed to predict tensile and compressive fibre failures, while the crack band model [8] is used to mitigate mesh sensitivity. Puck’s criteria [9] is incorporated in order to predict intralaminar damage by checking for damage initiation on multiple potential fracture planes. This provides a rational approach to the degradation of material stiffness depending on whether failure is similar to delamination, transverse cracking, or mixed mode. Although this does not explicitly consider interlaminar damage, it is a highly
efficient way to include the effects of transverse damage, similar to delamination, without resorting to the application computationally expensive cohesive elements at each ply boundary [10]. This material model was implemented in an Abaqus® UMAT subroutine [6]. A more thorough explanation of the implementation has been presented by Zhou et al. [10]. The elastic and damage development law properties employed are presented per Tables 3 & 4 respectively.

Table 3: Elastic properties of the composite material, HTA/6376 [11].

<table>
<thead>
<tr>
<th>$E_{11}$ (GPa)</th>
<th>$E_{22}$ (GPa)</th>
<th>$E_{33}$ (GPa)</th>
<th>$G_{12}$ (GPa)</th>
<th>$G_{13}$ (GPa)</th>
<th>$G_{23}$ (GPa)</th>
<th>$\nu_{12}$</th>
<th>$\nu_{13}$</th>
<th>$\nu_{23}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>139.3</td>
<td>10.14</td>
<td>10.14</td>
<td>6.02</td>
<td>6.02</td>
<td>3.9</td>
<td>0.32</td>
<td>0.32</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Table 4: CDM properties of the composite material, HTA/6376 [11].

<table>
<thead>
<tr>
<th>$X_T$ (MPa)</th>
<th>$X_C$ (MPa)</th>
<th>$Y_T$ (MPa)</th>
<th>$Y_C$ (MPa)</th>
<th>$S_{12}$ (MPa)</th>
<th>$S_{13}$ (MPa)</th>
<th>$a$</th>
<th>$b$</th>
<th>$R_0$ (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2170</td>
<td>1600</td>
<td>70</td>
<td>250</td>
<td>82.62</td>
<td>120</td>
<td>0.397</td>
<td>0.493</td>
<td></td>
</tr>
<tr>
<td>$Y_{12}$ ($\sqrt{\text{Pa}}$)</td>
<td>$Y_{12}$ ($\sqrt{\text{Pa}}$)</td>
<td>$Y_{22}$ ($\sqrt{\text{Pa}}$)</td>
<td>$Y_{22}$ ($\sqrt{\text{Pa}}$)</td>
<td>$R_0$ (MPa)</td>
<td>$\beta$ (MPa)</td>
<td>$\mu$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>48</td>
<td>3000</td>
<td>100</td>
<td>3100</td>
<td>21.59</td>
<td>512.9</td>
<td>0.41</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

There were two approaches adopted to model the adhesive layer. In order to analyse deformation and stress in the adhesive prior to the onset of damage, it was represented by an elastic-plastic constitutive model. Similar high-strength, two-part epoxy adhesive systems to that employed herein have been shown to exhibit dependence of yield on hydrostatic stress [12]. The Mohr-Coulomb yield criterion has been applied in order to capture this behaviour. The friction angle, $\phi$, which determines the influence of normal stress on the yielding behaviour of the material, and the cohesion yield stress, $c$ were assigned similarly to O’Dwyer et al. [12]. Correspondingly, non-associated flow was assumed, and the angle of dilation was set to zero. The mechanical properties of the adhesive system are summarised in Table 5. This material was discretised by 8-node, linear, brick elements with reduced integration and hourglass control (C3D8R) [6], with six elements through the thickness of the adhesive.

Table 5: Mechanical properties of the adhesive, Hysol 9466 [5, 12].

<table>
<thead>
<tr>
<th>$E$ (GPa)</th>
<th>$G$ (MPa)</th>
<th>$\nu$</th>
<th>$\sigma_{uts}$ (MPa)</th>
<th>$c$ (MPa)</th>
<th>$\phi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.718</td>
<td>636.6</td>
<td>0.35</td>
<td>32</td>
<td>30</td>
<td>15°</td>
</tr>
</tbody>
</table>

Subsequently, a mixed-mode cohesive zone damage model (CZDM) was adopted in consideration of both normal (mode I) and tangential (mode II & III) deformation and damage within the adhesive layer. The model is implemented in Abaqus® 6.14 [6]. The adhesive layer was represented, with its true thickness, by a single layer of 8-node cohesive elements (COH3D8) [6]. In each mode the response of these elements obey a bi-linear traction-separation law with linear
softening. The initial stiffness of the cohesive elements is governed by the elastic properties of the adhesive. Interaction between each mode of loading during damage initiation is accounted for according to the quadratic stress criterion, as per Eq. 2.

\[
\left( \frac{t_n}{t_o} \right)^2 + \left( \frac{t_s}{t_o} \right)^2 + \left( \frac{t_t}{t_o} \right)^2 = 1
\]

(2)

where \( t_n, t_s \) and \( t_t \) represent the normal, in-plane tangential and out-of-plane tangential tractions respectively, and \( t_o^n, t_o^s \) and \( t_o^t \) are the critical traction values in each mode.

Once damage has initiated, the stiffness of the cohesive elements softens progressively according to a damage parameter, \( D \), which monotonically evolves from 0 to 1 upon further loading after damage initiation. Damage evolution is controlled by the linear fracture energetic criterion, according to Eq. 3. Once this criterion is satisfied \( D \) is set to 1, the element may no longer carry load, and is deleted from the model, allowing fracture to propagate.

\[
\left( \frac{G_n}{G_C^n} \right)^\alpha + \left( \frac{G_s}{G_C^s} \right)^\alpha + \left( \frac{G_t}{G_C^t} \right)^\alpha = 1
\]

(3)

where \( G_n, G_s \) and \( G_t \) represent the normal, in-plane tangential and out-of-plane tangential fracture energy release rates respectively, \( G_C^n, G_C^s \) and \( G_C^t \) are the critical fracture energies in each mode, and \( \alpha = 1 \) for the linear fracture energetic criterion.

The properties of the CZDM were determined through an inverse calibration procedure. The tangential, mode II and mode III properties were assumed to be equal to one another, i.e. \( t_o^s = t_o^t \), and \( G_C^s = G_C^t \). The associated material properties are outlined in Table 6.

| Table 6: Cohesive zone model properties of the adhesive, Hysol 9466. |
|-----------------|-----------------|-----------------|-----------------|
| \( t_o^n \) (MPa) | \( t_o^s \) (MPa) | \( G_C^n \) (J/m²) | \( G_C^s \) (J/m²) |
| 32.8             | 18.6            | 0.44            | 0.687           |

By applying the CZDM in combination with the composite CDM, it was possible for the mechanical response of the interlocking joints to be accurately represented until catastrophic failure of the joint. Implicit simulations were conducted with the large strain formulation, NL-GEOM [6], and the specimen was loaded in quasi-static, displacement controlled tension from the grip locations shown in Figure 1a.

4 RESULTS & DISCUSSION

The simulated mechanical, load-displacement response of the interlocking joint configurations, C1, C2, and C3, are compared to representative experiments in Figure 2. The stiffness and strength of the simulations correspond well to that of the experiments for each configuration. The interlocking joints present an improvement in ultimate failure load compared to the standard joint; the most significant improvement being from C1. The magnitude of this increase is 18.3% based on the mean experimental results and 11.2% based on the corresponding
Simulations. This improvement is brought about as the interlocking surfaces inhibits relative
displacement between the male and female adherends during deformation. Interestingly, an in-
terlocking design similar to C2 (Figure 2b) demonstrated the most compelling improvements in
performance, by a significant margin, for a metal-metal interlocking adhesive joint configura-
tion [1]. However, in the hybrid joint configuration (C2), damage succumbed by the composite,
female adherend resulted in premature failure. In order to establish an understanding of the me-
chanics governing deformation and initial failure of the interlocking joint, the stress distribution
in the adhesive layer prior to damage propagation is considered in Section 4.1 and its damage
tolerance will be discussed in Section 4.2. Interlocking joint configuration 2 is considered in
detail; thorough analysis of configurations 1 and 3 remains for future work.

Figure 2: Mechanical response of simulated interlocking joint configurations, C1, C2, and C3,
compared to that of representative experimental specimens (a-c); and the ultimate failure loads
of the simulations compared to experimental results for each configuration (d).

4.1 Adhesive Stress Distribution

The adhesive was modelled as an elastic-plastic material, as per Section 3. A constant load,
$F$, was applied to the joint; two load levels were considered: 250N and 625N. These loads
corresponded to linear and non-linear deformation of the joint respectively, the latter being just
prior to failure, as determined from experiments. The normal and tangential adhesive stress
distributions were extracted at the centre of the adhesive thickness, in a path at the x-z mid-
plane of the joint. The stress distributions in the interlocking joint (C2) are compared to that
of the standard adhesive joint in Figure 3. In order to discuss the influence of the interlocking
morphology, the joint overlap is described by its female end, i.e. the end of the bond closest to
the constraint of the female (composite) adherend, and vice versa, its male end. The cutaway
cross-sectional profiles of each adhesive layer are also provided above Figure 3 for clarity.
Considering elastic deformation of the joint (F=250N), the normal stress distribution in the adhesive (Figure 3a) demonstrates concentrations at the ends of the joint overlap, with minimal difference between the interlocking and standard joints. However, in the interlocking joint there are also peaks in normal stress on the faces of the interlocking profile near both the female and male ends of the bond (i & ii); the peak at the female end (i) being greater as a result of the loading direction. This indicates that the mechanical interlock is relatively inactive for load transfer while the joint deforms elastically. However, there is also an increase in the compressive normal stress along the base of the interlocking profile (iii). The remainder of the bond area shows a similar stress distribution to that of the standard adhesive joint. In terms of the tangential stress distribution (Figure 3b), differences between the interlocking and standard joints in the vicinity of the interlocking profile are more apparent. Tangential stress at the female and male ends of the bond, around the profile, are increased and decreased respectively, compared to the standard joint. There is minimal stress in the adhesive along the base of the interlocking profile (iv).

During non-linear deformation of the joint (F=625N), the stress distribution in the adhesive evolves considerably. Normal stress in the adhesive (Figure 3c) at the profile face at the female end of the bond (i) has reached the ultimate tensile strength of the adhesive, indicating that failure would initiate in this region. Concentrations in stress at the ends of the overlap have become greater than that of the standard joint. But, compressive normal stress at the base of the
profile (iii) has also increased. In consideration of the tangential stress distribution (Figure 3d), as a result of plastic deformation of the adhesive at the ends of the joint overlap, the peak in shear stress moved towards the centre of the bond as the load increased, for both the interlocking and standard joints. However, for the interlocking joint, tangential stress at the female end of the bond (v) remains considerably greater than that at the male end. As a result, plasticisation of the adhesive at the female of the bond is more significant and the peak in stress has moved further, approaching the face of the interlocking profile. This indicates that failure initiates at the female end of the overlap when this peak in tangential stress reaches the face of the profile, where a concentration in normal stress has been identified, (i) in Figure 3.

4.2 Progressive Damage Analysis

The mechanical response and failure of interlocking joints C1 and C3 were similar. During experiments fracture initiated in the adhesive at the female end of the joint overlap and subsequently at the male end before propagating rapidly across the adhesive bond. At this point the male and female adherends popped out of their interlocked position as a result of insufficient lateral constraint and possibly due to elastic snap-back from the mechanical grips, given the sudden, large drop in load indicated by the simulation. Thus, the experimental specimens (C1 and C3) ultimately failed cohesively; although, the simulations indicated the potential for further loading and damage to the composite adherend. Experimentally, C2 demonstrated more interesting damage to the composite adherend. Although damage also initiated in the adhesive at the female end of the overlap, the joint ultimately experienced similar to shear-out or net-tension type failure in the composite adherend, as shown per Figure 4.

The simulation of C2 demonstrated similar mechanical behaviour to the experiment, being unstable prior to achieving its ultimate failure load, which was correspondingly reduced (Figure 2). It was observed from the simulation that, in fact, the adhesive on the profile face at the female end of the bond failed before the adhesive at the ends of the joint overlap, as similarly indicated by the adhesive stress analysis, per Section 4.1, and shown at point (i) in Figure 5a. Simultaneously, intralaminar, matrix damage initiated in the 90° ply at the base of the profile in
the composite adherend, point (ii), Figure 5a. This resulted in non-linear mechanical response of the joint; this is also the region in which damage of the composite adherend was first noted during the experimental tests (Figure 4). Once damage initiated in the adhesive at the profile face at the female end of the bond, damage began to develop in the adhesive at the female end of the overlap, point (iii) in Figure 5b. Matrix damage at point (ii) propagated into a second 90° ply and damage also initiated at another location at the base of the profile in the composite adherend, point (iv) shown in Figure 5b.

Figure 5: Contour plots of interlocking joint C2, in a section at the x-z mid-plane, showing the development of damage during various stages of loading, labelled (a)-(d).

As the joint achieved its ultimate failure load, significant damage developed in the adhesive at the female end of the overlap and fracture subsequently initiated in this region, point (v) shown in Figure 5d. The finite element model failed to converge at this point; where it is clear from the experimental test that the joint experienced a large drop in load (which was also indicated by the simulation), while damage developed in multiple regions of the composite adherend. Nonetheless, up to this point the simulation predicted the mechanical response of the joint as well as the development of intralaminar damage with reasonable accuracy. It may be seen from point (vi) of Figure 5 that the simulation also predicted matrix damage which resulted in the shear out of the composite adherend. However, given that it was not possible to run the simulation to completion, it is not yet clear that the simulation would accurately predict interlaminar damage that ultimately led to the failure of the joint. There was no fibre damage apparent from either the simulation or the experimental specimens. It was apparent that matrix damage initiated in and propagated through the 90° plies at the centre of the laminate (corresponding to the base of the interlocking profile). This highlights that the present laminate stacking sequence may be ineffective for the interlocking adhesive joint concept; 90° plies should not be included in the centre of the laminate, near the base of the interlocking profile. The incorporation of ±45° plies may also present reinforcement around the profile in the female
adherend in order to prevent the intralaminar failures which resulted in shear-out type failure. It should also provide better resistance to interlaminar damage.

5 CONCLUSIONS

The mechanical response of the interlocking adhesive joint concept has been investigated thoroughly through finite element analysis. The accuracy of the FE model was validated against experimental results, showing strong correlation in terms of stiffness, strength. The model also accurately predicts the progression of damage in the adhesive and the composite adherend, resulting in catastrophic failure of the joint. The concept demonstrates improvements of up to 18.3% in terms of the joint’s ultimate failure load, when compared to a standard adhesively bonded joint. Importantly, the concept also exhibits a shift in failure mode, from cohesive failure, to composite net-tension or shear-out type failure mode, similar to what may be observed for mechanically fastened joints. These failure modes can have significantly better damage tolerance than adhesive failure. It is apparent that the failure mode of the joint can be influenced through the design of the interlocking morphology. This development has the potential to mitigate a primary weakness of the adhesive joint: the rapid, catastrophic un-zipping of the adhesive bond that occurs during fracture. The mechanical performance improvements demonstrated by the rudimentary interlocking adhesive joint designs presented herein will be further enhanced through optimisation of the concept. In addition, it has been shown that the laminate stacking sequence employed was ineffective for the present interlocking design and should be revised to yield further improvements in performance. This will be the subject of future work.

Acknowledgements

The authors acknowledge the financial support of Science Foundation Ireland under Grant No. 13/IA/1833 and the Irish Research Council (IRC) for funding under their Government of Ireland Postgraduate Scheme through Grant No. GOIPG/2014/945. The authors also wish to acknowledge the DJEI/DES/SFI/HEA Irish Centre for High-End Computing (ICHEC) for the provision of computational facilities and support. The authors also acknowledge Mark O’Brien for the provision of the experimental data.

References


PREDICTION OF DAMAGE PROGRESSION IN NOTCHED TENSILE SPECIMENS: COMPARISON BETWEEN TWO INTRALAMINAR DAMAGE MODELS

Johannes Reiner*, Navid Zobeiry*, Reza Vaziri†, Thomas Feser†, Matthias Waimer†, Dominik Schueler†, Nathalie Toso-Pentecôte†

*Composites Research Network, The University of British Columbia
Vancouver BC, Canada
Reza.Vaziri@ubc.ca

†Institute of Structures and Design, German Aerospace Center (DLR)
Stuttgart, Germany
Matthias.Waimer@dlr.de

Key words: Simulation, Continuum Damage Mechanics, Notched Geometries.

Summary: In this work, two intralaminar damage models with different underlying assumptions are utilized to predict the damage response of notched composite laminates. One is a sub-laminate based continuum damage model, CODAM2, implemented in LS-DYNA, and the other is a ply-based damage model, ABQ_DLR_UD, that uses the Ladevèze damage model as its basis and is implemented as a user-material model (VUMAT) in Abaqus/Explicit. Aside from the base-formulations, different variations of the models including local and nonlocal integration (smearing) schemes are also considered. These models are employed to predict the damage response of Over-Height Compact Tension (OCT) and scaled center-notched specimens made out of IM7/8552 CFRP laminates with quasi-isotropic layup. For each geometry and loading scenario, sensitivity of the predictions with respect to various damage parameters including peak stress and fracture energy as well as assumptions made on the shape of the strain-softening curve are investigated. The results shed light on the effect of various assumptions in these two models on predicting damage progression in notched composite specimens.

1 INTRODUCTION

High performance carbon-fibre reinforced composite laminates are being used in a wide range of industrial applications. The most prominent field is the aerospace industry where composites have become the material of choice for the design of main load carrying components for instance in Boeing’s 787 or the Airbus A350.

In order to reduce physical testing and to optimize the design in a cost-effective manner, virtual testing of composite structures is essential. Despite tremendous efforts over the past 30 years, the development of robust and computationally efficient damage models is an ongoing and active area of research. Failure in heterogeneous and anisotropic composites occurs on multiple length scales with complex interaction between various failure modes which significantly increases the complexity of numerical analyses. Hence many computational models have been proposed in the literature to simulate the nonlinear structural response on various length scales and through different modelling concepts ranging
from fully discrete approaches to smeared continuum models [1, 2, 3].

The most popular approach to simulate damage in laminated composites is the finite element modelling at the ply-level combined with Continuum Damage Mechanics (CDM) [4, 5] due to the convenience of their implementation. Mesoscopic ply-based models are composed of several orthotropic layers of elements representing individual composite plies. In CDM, the mechanical properties of damaged material in the fracture process zone are smeared out by associating damage mechanisms with their overall effects on the mechanical material parameters.

We present two popular CDM approaches on the mesoscale applied to tensile loading of notched quasi-isotropic laminates made of IM7/8552 unidirectional CFRP layers. Since the focus of this numerical study is on intralaminar damage, only dispersed-ply laminates are considered where their response is less dominated by interlaminar damage effects such as delamination compared to blocked-ply laminates. The direct comparison between CODAM2 [1] developed at the University of British Columbia (UBC) and the Ladevèze-based damage model [6] implemented in Abaqus as ABQ_DLR_UD model [7] by the German Aerospace Center (DLR) shows the effect of two different CDM models applied to identical loading geometries, finite element meshes and boundary conditions.

This paper is structured as follows: Section 2 presents a brief overview of the two CDM approaches used to simulate intralaminar damage in the longitudinal and transverse directions of a ply. In Section 3, common and specific material parameters for modelling of IM7/8552 composites are presented. The models are applied to over-height compact tension and center-notched specimens under tension. Results are shown in Section 4 and the two damage models, CODAM2 and ABQ_DLR_UD, are quantitatively and qualitatively compared by evaluating the global force-displacement response and crack growth in the fracture process zone. Section 5 concludes the paper by providing a summary of the major findings and outlines future activities within the UBC/DLR collaboration.

2 INTRALAMINAR DAMAGE MODELS

In the following, the main concepts of each of the two models are outlined. Table 1 summarizes the variables used to describe each damage model.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_\alpha$</td>
<td>Elastic modulus</td>
</tr>
<tr>
<td>$G_{12}$</td>
<td>Shear modulus</td>
</tr>
<tr>
<td>$\nu_{12}$</td>
<td>Major Poisson’s ratio</td>
</tr>
<tr>
<td>$\nu_{21}$</td>
<td>Minor Poisson’s ratio</td>
</tr>
<tr>
<td>$\sigma_{11}$</td>
<td>Longitudinal stress</td>
</tr>
<tr>
<td>$\sigma_{22}$</td>
<td>Transverse stress</td>
</tr>
<tr>
<td>$\tau_{12}$</td>
<td>Shear stress</td>
</tr>
<tr>
<td>$\varepsilon_{11}$</td>
<td>Longitudinal strain</td>
</tr>
<tr>
<td>$\varepsilon_{22}$</td>
<td>Transverse strain</td>
</tr>
<tr>
<td>$\varepsilon_{22}^{eq}$</td>
<td>Equivalent stress</td>
</tr>
<tr>
<td>$\varepsilon_{\alpha}^{eq}$</td>
<td>Equivalent strain</td>
</tr>
<tr>
<td>$\sigma_{\alpha}^{eq}$</td>
<td>Equivalent stress</td>
</tr>
<tr>
<td>$\varepsilon_{\alpha}^{eq}$</td>
<td>Equivalent strain</td>
</tr>
<tr>
<td>$Y_{T}^{eq}$</td>
<td>Elastic shear strain</td>
</tr>
<tr>
<td>$Y_{20}$, $Y_{120}$</td>
<td>Damage initiation parameter</td>
</tr>
<tr>
<td>$Y_{2c}$, $Y_{12c}$</td>
<td>Damage evolution parameter</td>
</tr>
<tr>
<td>$Y_{2s}$, $Y_{12s}$</td>
<td>Brittle failure parameter</td>
</tr>
<tr>
<td>$X_{T}$</td>
<td>Longitudinal ply strength</td>
</tr>
<tr>
<td>$Y_{T}$</td>
<td>Transverse ply strength</td>
</tr>
<tr>
<td>$S_{L}$</td>
<td>Shear ply strength</td>
</tr>
<tr>
<td>$R_0$</td>
<td>Yield stress</td>
</tr>
<tr>
<td>$\beta$, $m$</td>
<td>Hardening coefficients</td>
</tr>
<tr>
<td>$Y_{20}$, $Y_{120}$</td>
<td>Damage initiation parameter</td>
</tr>
<tr>
<td>$Y_{2c}$, $Y_{12c}$</td>
<td>Damage evolution parameter</td>
</tr>
</tbody>
</table>

$\alpha = 1$: Longitudinal direction, $\alpha = 2$: Transverse direction, $\alpha = 12$: Shear direction

Table 1: Nomenclature for the variables used to describe the two intralaminar damage models.
2.1 CODAM2

The UBC Composite Damage Model, CODAM, introduced by Williams et al. [8], was originally formulated as a sub-laminate based continuum damage model. The second generation of this model, CODAM2, addressed the material objectivity issue of the original formulation and incorporated the nonlocal averaging feature which not only enabled mesh size and orientation invariant numerical results but also led to a more accurate prediction of the damage growth paths.

Here, the mesoscopic form of CODAM2, recently developed by Shahbazi et al. [9, 10] is used to simulate damage initiation and propagation at the ply level. Accordingly, damage initiation in a lamina is based on a simple maximum stress criterion and Hashin’s failure criterion [11] in the longitudinal (fibre) and transverse (matrix) direction, respectively. Fibre damage initiation function,

\[ F_1 = \frac{\sigma_{11}}{X_T} \]  

and damage initiation function in the transverse direction, \( F_2 \), accounts for the interaction of tensile and shear stresses such that

\[ F_2 = \left( \frac{\sigma_{22}}{Y_T} \right)^2 + \left( \frac{\tau_{12}}{S_{L}} \right)^2. \]

Damage is initiated when \( F_\alpha \geq 1 \), with \( \alpha = 1,2 \).

The equivalent strain component in the fibre direction is defined as

\[ \varepsilon_{1e}^{eq} = |\varepsilon_{11}| \]

and the equivalent transverse strain and stress are

\[ \varepsilon_{2e}^{eq} = \sqrt{\left(\varepsilon_{22}^{e} \right)^2 + \left(\varepsilon_{12}^{e} \right)^2} \quad \text{and} \quad \sigma_{2e}^{eq} = \frac{\left(\sigma_{22} \varepsilon_{22} + \tau_{12} \varepsilon_{12} \right)}{\sqrt{\left(\varepsilon_{22}^{e} \right)^2 + \left(\varepsilon_{12}^{e} \right)^2}}. \]

Damage variables \( \omega_\alpha, \alpha = 1,2 \) are formulated in terms of initiation and saturation strains \( \varepsilon_\alpha^i \) and \( \varepsilon_\alpha^s \) as illustrated in Figure 1(a) and (b) and are defined as

\[ \omega_\alpha = \frac{\varepsilon_\alpha^{eq} - \varepsilon_\alpha^i}{\varepsilon_\alpha^s - \varepsilon_\alpha^i} \quad \varepsilon_\alpha^s = \frac{2g_1}{T}, \quad \varepsilon_\alpha^i = \frac{2g_1}{X_T}, \quad \tau_\alpha = \sigma_\alpha^{eq} \Rightarrow F_\alpha = 1. \]

The stiffness reduction factors applied to the longitudinal, transverse and shear moduli, \( R_1, R_2 \) and \( R_{12} \), respectively, are given by

\[ R_\alpha = (1 - \omega_\alpha), \alpha = 1,2 \quad \text{and} \quad R_{12} = R_1 R_2. \]

Under plane-stress conditions, the secant stiffness matrix of the \( k^{th} \) ply, \( Q_k \), can then be written as
\[ Q_k = \frac{1}{D} \begin{bmatrix} R_1 E_1 & R_1 R_2 v_{12} E_2 & 0 \\ sym & R_2 E_2 & 0 \\ & D R_{12} G_{12} & \end{bmatrix}, \quad D = 1 - R_1 R_2 v_{12} v_{21}, \quad (8) \]

where the shear modulus \( G_{12} = G_{12}(\gamma_{12}) \) is a nonlinear function of the shear strain.

**Nonlocal Averaging**

In the nonlocal damage formulation, the stress state at a point is not only dependent on the local strain state but also on the state of strain in a finite neighborhood of that point [12]. In CODAM2 the equivalent strains \( \varepsilon_{eq}^\alpha \) at a point \( X_i \) are averaged over a spherical zone of volume \( \Omega_r \) such that

\[ \varepsilon_{eq}^\alpha (X_i) = \frac{1}{W} \int_{\Omega_r} \varepsilon_{eq}^\alpha (X) W(||X_i - x||) d\Omega \quad (9) \]

where \( W \) is a weight function and \( ||X_i - x|| \) a norm measuring the distance between the points \( X_i \) and \( x \) [1].

The zone \( \Omega_r \) should cover at least two integration points. Hence for reduced integration methods with one integration point per element, the radius \( r \) of the spherical or circular zone \( \Omega_r \) should be at least two times the characteristic element size.

**2.2 ABQ_DLR_UD**

The ABQ_DLR_UD damage model which uses the Ladevèze CDM ply model [6] has been implemented as an Abaqus/Explicit user material with minor modifications [7]. Here, the model was extended by applying the Bažant’s crack band theory [13] to capture energy dissipation due to fibre damage.

Fibre damage is initiated following a maximum stress criterion, \( F \geq 1 \), where

\[ F = \frac{\sigma_{11}}{X_T} \quad (10) \]

Fracture mechanics based fibre damage evolution is captured by introducing the scalar fibre damage parameter \( d_1 \) as a function of fibre strain in Figure 1(a) such that

\[ d_1 = 1 - \varepsilon_1^i \exp(\phi(\varepsilon_1^i - \varepsilon_{11})) \quad (11) \]

with \( \varepsilon_1^i \) being the fibre strain at damage initiation and \( \phi \) the softening coefficient given by

\[ \phi = \frac{2(1-k)}{2g^f_1 \varepsilon_1^i} \quad (12) \]

Here \( k \) is a small number that denotes fibre damage saturation according to \( \sigma_{11}(\varepsilon_1^s) = kX_T \).

For strains \( \varepsilon_{11} \geq \varepsilon_1^s \) the contribution to the dissipated damage energy can be neglected [14].

Transverse matrix and shear damage are captured by Ladevèze theory [2] with the damage functions in Figure 1(c) defined as

\[ d_2 = \frac{(Y - Y_{20})_+}{Y_{2c}} \quad \text{if} \quad d_2 < d_{max}, Y < Y_{12s} \text{ and } Y_2 < Y_{2s} \quad (13) \]

\[ \text{else} \quad d_2 = d_{max}, \]

\[ \]
\[ d_{12} = \frac{(Y - Y_{120})_+}{Y_{12c}} \quad \text{if} \quad d_{12} < d_{\text{max}}, Y < Y_{12s} \quad \text{and} \quad Y_2 < Y_{2s}; \]
\[ \text{else} \quad d_{12} = d_{\text{max}}. \]  

Based on experimental findings, Ladevèze introduced the quantities \( Y \) and \( Y_2 \) which are functions of associated forces \( Y_{12} \) and \( Y_2 \). Associated forces are analogous to energy release rates in crack propagation and govern damage development:

\[ Y = \sup_{\tau \in \sigma} (\sqrt{Y_{12}(\tau)} + bY_2(\tau)) \quad Y_2 = \sup_{\tau \in \sigma} (\sqrt{Y_2(\tau)}) \]  

The failure threshold parameters \( Y_{2s} \) and \( Y_{12s} \) control brittle failure where damage values reach their maximum value \( d_{\text{max}} \) close to or equal to 1. Progressive damage evolution is governed by \( Y_{20}, Y_{2c}, Y_{120}, Y_{12c} \) and the coupling parameter \( b \).

With \( R_\alpha = (1 - d_\alpha) \) and \( \alpha = 1, 2, 12 \) the secant stiffness matrix of the \( k^{th} \) ply, \( Q_k \), can then be written as

\[ Q_k = \frac{1}{D_\nu} \begin{bmatrix} R_1 E_1 & R_2 v_{12} E_2 & 0 \\ R_1 v_{21} E_1 & R_2 E_2 & 0 \\ 0 & 0 & D_\nu R_{12} G_{12} \end{bmatrix}, \quad D_\nu = 1 - v_{12} v_{21}. \]  

A plasticity law was introduced to model permanent transverse and shear strains due to
internal friction with the elastic domain function \( f \), the effective stresses \( \bar{\sigma}_{12}, \bar{\sigma}_{22} \), the plasticity coupling parameter \( \alpha \) and the hardening law \( R(p) \) with \( p \) being the equivalent plastic strain:

\[
f = \sqrt{\bar{\sigma}_{12}^2 + \alpha^2 \bar{\sigma}_{22}^2} - R(p) - R_0, \quad R(p) = \beta p^m.
\] (17)

3 MATERIAL PARAMETERS

A common set of input parameters is used for both simulations using CODAM2 and ABQ_DLR_UD. Unidirectional elastic properties for Hexcel HexPly IM7/8552 with nominal ply thickness 0.125 mm are given in Table 2 [15].

<table>
<thead>
<tr>
<th>Material</th>
<th>( E_1 ) (GPa)</th>
<th>( E_2 ) (GPa)</th>
<th>( G_{12} ) (GPa)</th>
<th>( \nu_{12} )</th>
<th>( X_T ) (MPa)</th>
<th>( Y_T ) (MPa)</th>
<th>( S_L ) (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>IM7/8552</td>
<td>165</td>
<td>9</td>
<td>5.6</td>
<td>0.34</td>
<td>2560</td>
<td>73</td>
<td>90</td>
</tr>
</tbody>
</table>

Table 2: Unidirectional elastic material parameters.

Both intralaminar damage models require the longitudinal (fibre) fracture energy, \( G^f_1 \), as input. Assuming that fibre damage occurs as a local event, the fibre fracture energy density \( g^f_1 \) can be calculated by considering a characteristic element length \( l_e \) such that \( g^f_1 = \frac{G^f_1}{l_e} \).

Due to the variation of reported fracture energies for IM7/8552 ranging from 81.5 [16], 94.1 [17], 112.7 [18] to 134.7 kJ/m\(^2\) [17], the following numerical study includes a range of values for \( G^f_1 \) from 80 to 160 kJ/m\(^2\) in increments of 40 kJ/m\(^2\).

Transverse Fracture Energy

The transverse fracture energy density \( g^f_2 \) in equation (6) determines the area under the softening curve. By considering the nonlocal averaging radius \( r \), \( g^f_2 \) can be related to the intralaminar fracture energy \( G^f_2 \) such that

\[
g^f_2 = \frac{G^f_2}{0.65r}.
\] (18)

Using CODAM2, a combined experimental-numerical calibration technique applied to blocked and dispersed cross-ply laminates of IM7/8552 CFRP, \([0/90]_2s\) and \([0/90]_{8s}\), respectively, yields a value of \( G^f_2 = 2.6 \text{ kJ/m}^2 \) that correctly captures matrix cracking and delamination [9, 10]. This is significantly higher than typical interlaminar fracture energies which range from 0.2 to 1.0 kJ/m\(^2\), but it is in line with experimental quantification of intralaminar fracture energy in a similar composite system [19].

In contrast, ABQ_DLR_UD does not follow a fracture mechanics approach for transverse failure and instead, a brittle failure is assumed once the failure criteria are met according to equations (13) and (14).

4 RESULTS & DISCUSSION

In the following, the two intralaminar damage models are applied to notched composite specimens and results are compared to corresponding experimental data. Differences between the two approaches with respect to their predictions of the global force-displacement
response and local damage behaviour are discussed.

4.1 Over-Height Compact Tension of \([45/90/-45/0]_{2S}\) Laminates

The Over-Height Compact Tension (OCT) [20, 21] loading geometry produces stable and self-similar crack growth. Figure 2 shows the geometry and dimensions of the OCT in combination with the meshed specimen. The element size in the expected fracture zone at the notch is 0.5 mm and hence \(l_e = 0.5\) mm. This is the value used to calculate the fibre fracture energy density as mentioned in Section 3.1. Displacement in vertical direction is applied at the loading pins (green) in Figure 2 (b). Based on identical finite element meshes, the ABQ_DLR_UD model uses a single layer of multilayered shell elements to model the \([45/90/-45/0]_{2S}\) IM7/8552 composite laminate, whereas CODAM2 is applied to eight layers of thick (continuum) shell elements stacked through the thickness.

In Figure 3, the resulting force is plotted against the Pin Opening Displacement (POD) predicted by CODAM2 and ABQ_DLR_UD for the three fibre fracture energy levels of 80, 120 and 160 kJ/m\(^2\). It can be seen that the application of the two intralaminar damage models lead to a similar response. CODAM2 predicts a slightly higher peak force compared to ABQ_DLR_UD, whereas post-peak behaviour shows higher forces in the Ladevèze-based approach. The simulations are also compared to experimental data [20]. Considering the absence of interlaminar capabilities in the two models, the onset of damage is predicted with reasonable accuracy. However, it is believed that the implementation of cohesive interfaces/elements will significantly improve the post-peak behaviour which is characterized by a plateau in force rather than the predicted decrease in Figure 3. Since the focus of this numerical study is on the intralaminar damage behaviour, future research will investigate the effect of cohesive interfaces on the global force response.
Figure 3: The two intralaminar damage models CODAM2 and ABQ_DLR_UD compared by the global force-POD response for different fibre fracture energies of [45/90/-45/0]_2S IM7/8552 composite laminates.

Damage of the [45/90/-45/0]_2S laminate in matrix and fibre is further studied in Figure 4. Since the numerical models only consider intralaminar damage, crack initiation and propagation are identical in each sublaminate [45/90/-45/0]. Figure 4 shows the damage length $a$ in every ply of a sublaminate predicted by CODAM2 and ABQ_DLR_UD for the three different fibre fracture energies. In order to compare the two models consistently, matrix cracking is evaluated at damage initiation, i.e. at $\omega_2 > 0$ and $d_2 > 0$. A crack in fibre direction is considered to propagate when $\omega_1 \geq 0.98$ and $d_1 \geq 0.98$ where the two fibre damage models intersect as indicated in Figure 1(a). The damage analysis shows that damage initiation in matrix and fibre directions coincides in both ABQ_DLR_UD and CODAM2. However, the subsequent behaviour differs where damage in both directions appears to propagate faster in CODAM2. By considering the applied fibre damage in Figure 1(a), CODAM2 reaches damage saturation state ($\omega_1 = 1$) earlier and hence damage in fibre direction propagates faster compared to ABQ_DLR_UD. With fibre damage being the dominating driving force, the differences in matrix damage evolution can be explained as a consequence of the earlier fibre propagation in CODAM2. Comparing the application of the fracture energy in CODAM2 in Equation (18) and sudden matrix failure in ABQ_DLR_UD, matrix damage evolution seems to be negligible in this case. The analysis in Figure 4 also explains the differences in the global force-POD response in Figure 3. The slightly faster damage growth in fibre direction in CODAM2 is reflected by a lower post-peak behaviour in Figure 3.
Figure 4: Evaluation of ply damage in transverse (matrix) and longitudinal (fibre) direction in OCT simulations by comparing the crack length $a$ in CODAM2 and ABQ_DLR_UD models for different fibre fracture energies.

4.2 Centre-Notched \([45/90/-45/0]_{4S}\) Laminates

The two damage models are also applied to the \([45/90/-45/0]_{4S}\) centre-notched specimen under tension [22] in Figure 5. As in previous section, the mesh size in the expected fracture zone around the notch is 0.5 mm. Similarly, based on identical finite element meshes, ABQ_DLR_UD uses a single layer of multilayered shell elements and CODAM2 is applied to stacked layers of thick shell elements.

Figure 5: Geometry and dimensions of the simulated centre-notched specimen.

Figure 6 shows the predicted stress-strain response of the two intralaminar models for the three fibre fracture energies. The results are consistent with previous findings in Figure 3 and 4. The application of CODAM2 yields slightly higher strength values compared to ABQ_DLR_UD. The uniformly applied load leads to sudden failure near the notch. The range of applied fibre fracture energies covers the band of experimental strength data [22].
Table 3 shows a quantitative comparison of the predicted tensile strength for the different fracture energies. The two intralaminar damage models are within 8% of each other.

![Stress-strain response comparison](image)

**Figure 6:** Overall stress-strain response of centre-notched specimen simulated by CODAM2 and ABQ_DLR_UD with different fibre fracture energies. Also shown are the experimental strength results (gray) [22].

<table>
<thead>
<tr>
<th>Method</th>
<th>Fibre Fracture Energy (kJ/m²)</th>
<th>Tensile Strength (MPa)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiments</td>
<td>508-527 [22]</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FE predictions</td>
<td>80</td>
<td>CODAM2: 497.40</td>
<td>4.75</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ABQ_DLR_UD: 473.78</td>
<td></td>
</tr>
<tr>
<td></td>
<td>120</td>
<td>CODAM2: 572.64</td>
<td>5.96</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ABQ_DLR_UD: 538.51</td>
<td></td>
</tr>
<tr>
<td></td>
<td>160</td>
<td>CODAM2: 639.28</td>
<td>7.99</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ABQ_DLR_UD: 588.17</td>
<td></td>
</tr>
</tbody>
</table>

Table 3: Comparison between predicted and experimental tensile strength in centre-notched laminates.

A qualitative comparison with experimental CT results [22] is shown in Figure 7(a). The damage zone obtained by CODAM2 in Figure 7(b) correlates well with the experimental damage zone of approximately 1mm x 1mm. It can be seen that fibre damage in CODAM2 is affected by the nonlinear matrix damage. In contrary, the purely local damage model in ABQ_DLR_UD yields a localized damage band. Note that splitting is visible in the experiment which is caused by interlaminar failure. It is expected that the addition of cohesive interfaces/elements in the numerical models will capture these effects accurately. However, the global stress-strain response in Figure 6 is not expected to change significantly. The brittle failure indicates that delamination has only a minor effect on the global response.
5 CONCLUSIONS

This work directly compares two popular Continuum Damage Mechanics (CDM) models applied to notched composite laminates under tension to study intralaminar damage initiation and progression. CODAM2 uses a mesoscopic CDM approach with nonlinear shear and nonlocal matrix damage. The ABQ_DLR_UD is based on the popular Ladevèze damage model. Despite different underlying assumptions, the two intralaminar models predict similar damage initiation and propagation as well as global force response in the OCT and centre-notched specimens. In addition, the comparison with the experimental OCT data shows that CDM models require the capability to account for interlaminar delamination in the form of cohesive interfaces or elements in order to realistically capture the laminate behaviour after damage initiation. By considering a range of fibre fracture energies, this study also shows the sensitivity of typical CDM approaches to this input parameter, regardless of the chosen damage model.

Future work will focus on the implementation of interlaminar damage models and the application to larger structures under dynamic loading. The collaboration between UBC and DLR will further contribute to highlight the effect of typical input parameters of commonly used CDM approaches such as CODAM2 and ABQ_DLR_UD. It will shift the focus of the choice of continuum damage model to the task of how to use existing CDM approaches effectively and successfully.

REFERENCES


DEVELOPMENT OF A NEW STANDARD FOR DYNAMIC MECHANICAL PROPERTIES CHARACTERIZATION OF CFRP

T. Schmack*, T.J.J. Jansen*, T. Filipe*, F. Walther†

*Light Weight Center, N/EK-114, Audi AG
NSU-Straße 1, 74148 Neckarsulm, Germany
tobias.schmack@audi.de

†Department of Materials Test Engineering (WPT), TU Dortmund University
Baroper Str. 303, 44227 Dortmund, Germany
frank.walther@tu-dortmund.com

Keywords: Strain rate, Material characterization, Compression test.

Summary: This study deals with the development of a new standard for strain rate-dependent characterization of carbon fibre-reinforced plastics. The characterization of these strain rate-dependent mechanical properties is essential to improve Finite Element Analysis to use the material at its full potential and reduce the overall weight and cost of the product.

Former research shows the strain rate dependency of carbon fibre-reinforced plastics for in-plane properties. There is however currently no standard available for this type of tests. Therefore a compression fixture is developed to perform dynamic tests on rectangular cross-section specimens. A study is performed to find the specimen geometry that leads to the optimal results. Using this optimal geometry, the strain rate-dependent in plane compressive strength and stiffness of a carbon-epoxy laminate is characterised. The new method is validated using experimental and numerical multi-scale validation.

Using this new developed testing standard, the testing results for ultimate strength and failure strain together with simulation results for ultimate strength modelling are strongly improved. Furthermore, insight is gained into the dependency of compression strength with respect to sample geometry.

1. INTRODUCTION

In the design process of load bearing components, the response to dynamic loads is an important design criterion. With increasing usage of fibre-reinforced composites for structural components across the automotive and aerospace industry, it became crucial to understand how these materials behave when subjected to different rates of strain [1]. This understanding enables the use of the material at its full potential and reduces the overall weight and cost of the product. In the case of a side impact pole crash for an Audi R8, the strain rate distribution of the carbon fibre reinforced plastics (CFRP) back wall ranges from $1 \text{s}^{-1}$ up to $100 \text{s}^{-1}$. Therefore Finite Element Analysis (FEA) crash simulations are performed. Optimizing prediction accuracy of the simulation, requires reliable material characteristics obtained with dynamic testing.
To determine the strain rate-dependent material characteristics, it is first analysed whether tension or compression tests need to be performed. Using a tensile Split Hopkinson Pressure Bar (SHPB), Harding and Welsh [2] performed dynamic tensile tests. They found no correlation between the tensile properties in fibre direction and the rate of loading. This is in agreement with tensile tests on carbon fibre bundles performed by Zhou et al. [3]. Using the SHPB testing method, Gilat et al. [4] performed transverse tension experiments on unidirectional carbon-epoxy composites at varying strain rates. This showed an increasing transverse modulus and failure stress with increasing strain rate, which was also observed by Taniguchi et al. [5]. For tension, this concludes that the strain rate dependency is only observed for off axis loading directions which is i.e. matrix dominated and therefore related to the strain rate dependency of the polymer matrix [6, 7]. Since the tensile properties of fibre-reinforced polymers loaded in fibre direction are fibre-dominated [3], it is comprehensible that the composite response at different strain rates is similar to the behaviour of carbon fibre bundles, i.e. not strain rate sensitive.

The longitudinal compressive strength of fibre-reinforced polymers in comparison to tensile strength is often significantly lower in axial loading direction since it is intimately controlled by the matrix behaviour [8]. When loaded axially in compression, the fibres tend to micro-buckle and, consequently, create kink bands [9, 10]. The compressive strength is therefore widely considered the more critical property and should thus be tested.


For dynamic compression testing of FRP there is however no adopted standard. Studies on strain-rate dependent compression testing show a large diversity in terms of testing methods and material combinations without facilitating a general conclusion for an ideal test method [1]. All studies do however show a significant strain rate dependence of the FRP. In a recent study performed by Koerber and Camanho [15], rectangular IM78552 carbon-epoxy specimens were tested using a compression SHPB apparatus. Koerber and Camanho reported a 40% increase in longitudinal compressive strength and a similar increase in failure strain at a moderate strain rate $\dot{\varepsilon} \approx 100 \text{s}^{-1}$ compared to quasi static conditions.

Concerning the rectangular specimens, different geometries are recommended by different testing methods [11, 13, 14]. Therefore it is important to understand the influence of each geometrical factor (tickness, width and gauge length) on the measured compression properties. Hsiao et al [16] found with the CLC test method an increase in compression modulus and a de-
crease in both compression strength and failure strain with increasing thickness. In more recent studies by Lee and Soutis [17], it is shown that for increasing thickness as well as increasing specimen volume, failure tends to occur at lower strain levels and thus leading to decreased compression strengths. This is also shown by Bing and Sun [10]. For off-axis compression testing of carbon-epoxy laminates, a significant decrease in compression strength with increasing specimen width is observed. The width should however be at least five times the specimen thickness to avoid the effect of free edges and reach an uniaxial state of stress [17].

The maximum gauge length for compression testing of FRP is limited by the phenomenon of Euler column buckling. The gauge length should however be long enough to place strain gauges and to allow stress decay to uniaxial compression, given the concentration caused by the grips end [11]. The magnitude of the stress concentration is caused by the stiffness mismatch at the end of the loading tabs. This depends on load introduction type, material of the loading tabs and its geometry (tapered or untapered). Regarding this geometry of the tabs, the most important factor is the taper angle at the grips end. Using tabs with a taper angle of $20^\circ$ can lead to reductions in stress concentrations up to 50%, when compared to untapered tabs [18].

The objective of this work is to develop a new methodology for strain rate-dependent characterization of CFRP as there is currently no adopted standard. Therefore a compression fixture is developed to perform dynamic tests on rectangular cross-section specimens. Taking the geometrical restraints of the fixture and load limitations of the testing machine into account, a study is performed to find the specimen geometry that leads to the optimal results. Using this optimal geometry, the strain rate-dependent in plane compressive strength and stiffness of a carbon-epoxy laminate is characterised.

In a first step of the multi-scale validation, these results are compared to the results obtained in quasi-static tests performed according to the standard ASTM D6641 [13] and DIN EN ISO 14126 [14]. In a second step, the experimentally obtained properties are used to simulate the laminate behaviour and compare this with the experimental results on coupon level. Finally, it is experimentally and numerically validated whether the strain rate dependency is observed beyond a coupon level and into more complex structural components using omega-profile cross-sectional components.

2. MATERIALS AND SPECIMEN PREPARATION

2.1 Materials

The specimens manufactured consist of DT120 epoxy matrix by DeltaTech reinforced with high-strength standard-modulus T700S carbon fibres by Toray. The unidirectional prepreg tape is 0.15 mm thick and has a fibre areal weight of 150 $g/m^2$. 


2.2 Rectangular cross-section specimens preparation

The plies were hand-laid as rectangular plates with dimensions of $500 \times 600 \text{mm}^2$ and cured in an autoclave for 1.5 hours at $120^\circ\text{C}$. Nine Fibre Volume Content (FVC) measurements were performed for each plate, resulting in an average FVC of 53.5% ($\pm 0.3\%$). The cured plates were then equipped with loading tabs made of quasi-isotropic Glass fibre-reinforced Plastic (GFRP) (2 mm thick) using DP490 Scotch-Weld epoxy adhesive by 3M. The individual specimens were cut to the final dimensions using a water-cooled rotating diamond saw. Specimens with three different layups were manufactured: $[0^\circ]_{16}$, $[90^\circ]_{16}$ and $[\pm 45^\circ/\pm 45^\circ]_{4s}$.

Norm specimen geometry

The specimens required for validating the newly developed method are according to the type B2 geometry which is defined by DIN EN ISO 14126 [14] and allowed by ASTM D6641. Both standards however do require untapered tabs, therefore a specimen set with untapered ($90^\circ$) tabs ("untapered Norm specimens") and a set with tapered ($30^\circ$) tabs ("tapered Norm specimens") is prepared for each layup.

2.3 Omega-profile cross-section specimens preparation

The second type of samples manufactured consists of 100 mm long omega-profile cross-section specimens. The prepreg system used is identical to the previously mentioned one. The individual unidirectional layers were hand-laid with a $[0^\circ/\pm 45^\circ/ + 45^\circ/ - 45^\circ/0^\circ/90^\circ]_s$ layup on a concave omega-shape mould and cured at $120^\circ\text{C}$ for 1.5 hours, resulting in the specimen dimensions shown in Figure 1 (after trimming).

![Figure 1. Omega cross-sectional component. a) Isometric view; b) Cross-section view](image)

3. EXPERIMENTAL SETUP

For the characterization of the compressive behaviour of carbon-epoxy laminates, strain rates varying from the typical quasi-static values (order of magnitude $10^{-5}$ s$^{-1}$) to moderate (up to 100 s$^{-1}$) are applied. Therefore, the static and dynamic tests were performed using an Instron Very High Speed (VHS) servo-hydraulic testing machine. For the first validation step on coupon level, compression tests were performed according to the standard at quasi-static speed using a Zwick Z100 testing machine.
3.1 Instron Very High Speed servo-hydraulic testing machine

The vertical shaft of this testing machine is displacement controlled. Using the same testing machine across the entire considered range of strain rates ensures that the effect of the fixture/machine on the measured results are constant.

Two fixtures were developed to clamp the two different specimen shapes. When designing the two fixtures, special attention was paid to ensure a similar effect on the specimens. Figure 2 shows the two fixture configurations: rectangular cross-section specimens (a) and omega-profile cross-section specimens (b). Both fixtures introduce the load via combined shear-end loading, reducing the stress concentration factor at the clamps. A piezoelectric load cell (Kistler 9051A for the rectangular cross-section specimen and Kistler 9071A for the omega-profile cross-section specimens) mounted on the fixture base, measures the load applied on the specimen.

Using two Photron FASTCAM SA-X high-speed cameras together with the Digital Image Correlation (DIC) software ARAMIS® supplied by Gesellschaft für optische Messtechnik (GOM), the specimen strain is measured.

3.2 Zwick Z100 testing machine

The validation tests on coupon-level are performed on a Zwick Z100 testing machine that uses the Hydraulic Composites Compression Fixture (HCCF) developed and patented by IMA Dresden. This fixture fulfills the requirements of DIN EN ISO 14126 Method 2 and ASTM D6641. A part of the compressive force is loaded via specimen clamping, the rest on the ends of the specimen. The force is measured using the Xforce K Zwick Roell loadcell. All the specimens tested according to the standard are equipped on both sides with linear strain gauges (Type 1-LY71-3/120, Hottinger Baldwin Messtechnik GmbH) in longitudinal direction.
4. EXPERIMENTAL RESULTS

4.1 Specimen geometry optimization

To determine the optimal specimen geometry, a 2-level full factorial design approach is followed with an additional investigation on the influence of a tab taper angle of a tapered (30°) or untapered tab (90°). The compression strength was chosen as the criterion to compare between different specimen geometries since the compression strength is affected by variations in both thickness and width [10, 16, 17]. This optimization shows that specimens with tapered tabs are able to support much higher loads due to smaller stress concentration factors at the tab ends. The average coefficient of variation in compression strength decreases from 14.4% to 5.6% when using tapered tabs instead of untapered tabs. This indicates that tapered specimens are less susceptible to small manufacturing defects or misalignments in the placement of the GFRP tabs. The optimal width is accurately predicted by a quadratic model at 12 mm width. However, no statistically significant difference in strength was found for different gauge lengths and for 2 mm or 4 mm thickness. A low thickness is however advised to prevent unacceptable failure modes as described in ASTM D6641 [13].

Using the Design of Experiments software MODDE® by the company Umetrics, the optimal geometry was found based on the probability of failure. This results in a final dimension (Figure 3) of 12 × 10 × 2.4 mm (width × gauge length × thickness) with tapered load introduction tabs of 35 mm long.

Using the Design of Experiments software MODDE® by the company Umetrics, the optimal geometry was found based on the probability of failure. This results in a final dimension (Figure 3) of 12 × 10 × 2.4 mm (width × gauge length × thickness) with tapered load introduction tabs of 35 mm long.

Figure 3. Rectangular optimized specimen geometry and dimensions.

4.2 Strain rate dependent material characterization

Using the optimized specimen geometry, specimens with [0°]₁₆, [90°]₁₆ and [+45°/ − 45°]₄ₛ layup were experimentally tested at the Instron VHS servo-hydraulic testing machine. These tests were performed at six different impactor velocities (which develops into specimen deformation speed), ranging from 2 mm/min to 1 m/s. For each combination of specimen layup and impactor speed, ten replicated experiments were performed to ensure an appropriate average accuracy.

Using the tapered and untapered Norm specimen geometry, specimens with [0°]₁₆, [90°]₁₆ and [+45°/ − 45°]₄ₛ layup were experimentally tested at the Zwick Z100 testing machine. These tests, called "Norm tests", were performed at a quasi-static speed of 2 mm/min. For each combination of specimen layup, ten replicated experiments were performed with the tapered Norm
specimens and five replicated experiments were performed with the untapered Norm specimens.

The strain rates for dynamic tests performed in fibre direction ([0◦]16 layup), ranged between $2.3 \times 10^{-4}$ and $4 \, \text{s}^{-1}$. Throughout this range, the longitudinal modulus does not show significant changes. On the other hand, the compression strength and fracture strain increase by 22% and 26%, respectively. Figure 4 depicts the average longitudinal stress-strain curves obtained at each of the six loading rates tested. Table 1 summarizes the in-fibre direction compression properties.

Figure 4 and Table 1 also show the results of the Norm tests performed in fibre direction with (1) untapered and (2) tapered Norm specimens. At quasi-static testing speed, a 35% increase in compression strength and a 42% increase in fracture strain is observed when using tapered tabs instead of untapered tabs. This clearly shows the detrimental effect of stress concentration on the analysis of compression strength and fracture strain, caused by the use of untapered tabs.

Comparing the quasi-static results obtained using the optimized specimen geometry with the results obtained by testing the tapered Norm specimens, shows that the newly developed fixture with optimized specimen geometry provides the same results. The average longitudinal stress-strain curves for the [90◦]16 and [+45◦/ − 45◦]4s layup, shown in Figure 5, indicate that tests performed with the optimized specimen geometry and newly developed fixture give an even better result than results obtained with the Norm tests. While performing the Norm test with the [90◦]16 layup, 80% and 30% of the tests with untapered and tapered tabs respectively were declared invalid due to the exceeded maximum bending factor of 10% [13, 14]. This concludes that the new developed method produces the same or even better results compared to the standard DIN EN ISO 14126 and ASTM D6641 where it is essential to apply tapered tabs.

In transverse direction ([90◦]16 layup), higher strain rates were achieved for dynamic tests with the optimized specimen geometry. These strain rates range between $2.9 \times 10^{-3}$ and $70 \, \text{s}^{-1}$. Both the transverse fracture strain and modulus are unaffected by the deformation rate, as they are kept more or less constant at 5.2% ± 0.18% and 8.4 GPa ± 0.07 GPa, respectively. Nonetheless, a significant increase in yield and fracture stress, as high as 56% for both these properties is observed. With increasing strain rate, the transverse response of the carbon-epoxy laminate becomes more brittle, with yielding and failure occurring at higher stress levels, as shown by the transverse stress-strain curves of Figure 5a.
Table 1. Averaged longitudinal compression properties (\([0^\circ]_{16}\) layup specimens).

<table>
<thead>
<tr>
<th>Strain Rate $\dot{\varepsilon} [s^{-1}]$</th>
<th>Strength [MPa]</th>
<th>Fracture Strain [%]</th>
<th>Longitudinal Modulus [GPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1) $2.2 \pm 0.1 \times 10^{-4}$</td>
<td>858 ± 62</td>
<td>0.84 ± 0.07</td>
<td>108.7 ± 2.1</td>
</tr>
<tr>
<td>(2) $2.1 \pm 0.1 \times 10^{-4}$</td>
<td>1158 ± 53</td>
<td>1.19 ± 0.07</td>
<td>109.4 ± 1.2</td>
</tr>
<tr>
<td>2.3 $\pm 0.1 \times 10^{-4}$</td>
<td>1165 ± 10</td>
<td>1.15 ± 0.03</td>
<td>104.9 ± 2.2</td>
</tr>
<tr>
<td>7.2 $\pm 0.5 \times 10^{-4}$</td>
<td>1215 ± 45</td>
<td>1.18 ± 0.06</td>
<td>112.8 ± 2.1</td>
</tr>
<tr>
<td>7.6 $\pm 0.4 \times 10^{-3}$</td>
<td>1288 ± 10</td>
<td>1.29 ± 0.04</td>
<td>106.9 ± 3.0</td>
</tr>
<tr>
<td>7.4 $\pm 0.5 \times 10^{-2}$</td>
<td>1352 ± 25</td>
<td>1.33 ± 0.05</td>
<td>110.1 ± 3.5</td>
</tr>
<tr>
<td>6.6 $\pm 0.4 \times 10^{-1}$</td>
<td>1380 ± 13</td>
<td>1.36 ± 0.05</td>
<td>111.8 ± 2.9</td>
</tr>
<tr>
<td>4.1 $\pm 0.5$</td>
<td>1422 ± 33</td>
<td>1.45 ± 0.06</td>
<td>111.2 ± 2.7</td>
</tr>
</tbody>
</table>

The shear properties are also affected by the deformation rate, as verified when testing \([+45^\circ/ - 45^\circ]_{4s}\) layup specimens in strain rates ranging from $6.2 \times 10^{-3}$ to $70 \, s^{-1}$. Similarly to the longitudinal and transverse moduli, the shear modulus does not seem to change with increasing rates. Nonetheless, the shear yield stress and strength (defined as the shear stress when the shear strain reaches 5\%) increase by 37\% and 27\%, respectively. Figure 5b shows the shear stress-strain curves obtained at the different strain rates tested together with the Norm test results.

Figure 5. Stress-strain curves for rectangular specimen.
Left: $[90^\circ]_{16}$ layup; Right: $[+45^\circ/ - 45^\circ]_{4s}$ layup.

(1) untapered Norm specimens; (2) tapered Norm specimens
4.3 Omega-profile cross-section specimens

The component tests with omega-profile cross-sectional specimens were performed with impactor velocities of $3.3 \times 10^{-5} \, m/s$, $0.01 \, m/s$ and $1 \, m/s$. For each speed, three replicated experiments were carried out.

With increasing deformation rate, the maximum cross section load sustained by the specimens increased from 55.6 kN to 67.3 kN, which represents an increase of 21%. This growth in strength is in line with the increase previously presented for rectangular specimens with a $[0^\circ]_{16}$ layup, as the $0^\circ$ plies are the main load-carrying plies of the omega-profile cross-section specimens. Figure 7 shows the compression force versus piston displacement for the nine specimens tested. It is relevant to note the inconsistency in impactor displacement at failure between the three testing velocities, which is related to fixture and testing machine limitations, allowing the specimens to slightly move inside the fixing jaws. For this reason, the impactor displacement after impact does not translate directly into specimen strain.

5. EXPERIMENTAL AND NUMERICAL MULT-SCALE VALIDATION

5.1 Validation on coupon-level

In a first validation step, the obtained results with the new developed fixture and optimized specimen geometry are compared to the Norm test results according to the DIN EN ISO 14126 and ASTM D6641 norm. These results show a strong improvement in obtained compression strength and fracture strain when using tapered tabs. Figure 6 shows the compression strength at different strain rates for tests performed in fibre direction using the new developed method compared to other testing fixtures and specimen geometry. This shows that the lowest compression strength is obtained when using thicker specimens with pure end loading. The results obtained using the new developed method ("Audi Axial Compression"), provide the highest compression strength and confirm that this matches the results obtained for testing tapered Norm specimens at quasi-static speed.

5.2 Validation on component level

To validate whether the strain rate dependency is also observed beyond a coupon level and into more complex structural components, the behavior of an omega cross-sectional component is experimentally and numerically analysed. Using the new developed method, the longitudinal Young’s modulus and rupture strain, transverse Young’s modulus, yield stress, strength and shear modulus are obtained on coupon-level. Using these obtained material characteristics, the material card in the commercially available explicit Finite Element software Pam-Crash from ESI is updated. As was experimentally verified, neither the longitudinal, transverse or shear moduli are affected by the deformation rate. Therefore, the ply behaviour can be modelled by using only two viscosity functions - also denoted functions of evolution.
Figure 6. Comparison between the new developed method, the standard and other external testing methods and specimen geometry [1].

Figure 7 shows the comparison between the experimentally and numerically obtained force-displacement graphs at three different strain rates. The experimental curves are linear until failure, where the cross-section force immediately drops to zero. This indicates that failure occurs due to material failure. In the simulation this is seen as elements being eliminated due to their longitudinal maximum allowed strain being reached. With increasing deformation rate, the failure displacement increases and the increase in simulated maximum load (+21\%) is identical to the experimental increase. This shows that strain rate dependent behavior is observed on component level and that the numerical model is able to predict the maximum with a relative error below 5\% for the three testing velocities considered.

Figure 7. Experimental and simulation force versus displacement curves for omega-profile specimens.
6. Conclusion

The objective of this work was to develop a new methodology for strain rate-dependent characterization of CFRP, as there is currently no adopted standard. A combined loading compression fixture is developed since the compressive strength is widely considered the more critical property and tensile properties in fibre direction show no strain rate dependency. With this fixture, a study on specimen optimization is performed. The use of tapered tabs reduces stress concentrations, this improves the obtained longitudinal compression strength with 35%. The obtained material characterization results are similar to the characteristic values obtained by the accepted standards DIN EN ISO 14126 and ASTM D6641 when tapered tabs are used.

Characterisation of the strain rate dependency of the T700-DT120 carbon-epoxy system shows a logarithmic increase of the longitudinal strength and failure strain with increasing strain rates, reaching growths up to approximately 25% for the range of deformation considered. An increase in transverse yield and ultimate stress of 56% was found over the respective quasi-static values, when tested at a strain rate of 70 s\(^{-1}\).

On component level, the strain rate effect is observed for omega-profile cross-section specimens with a multidirectional layup. Here, an increase in maximum force of 21% is registered. The on coupon-level obtained strain-rate dependent compression properties were implemented on a numerical model, resulting in satisfactory predictions of the experimental impact tests across the entire range of deformation rate considered.

This experimental and numerical multi-scale validation proves the validity of the new method.

References


