

UNIVERSITY OF PORTO
FACULTY OF ENGINEERING



A Continuum-Damage Model for Advanced Composites

Pedro Miguel Vieira Pinto Bandeira

Licensed in Mechanical Engineering by University of Porto,
Faculty of Engineering

A Thesis presented to the

Faculty of Engineering of the University of Porto

In partial fulfillment of the requirements for the Master of Science Degree in
Mechanical Engineering

Supervisor

Professor Doutor Pedro M. P. R. C. Camanho

Porto, August 14, 2005

Abstract

Composite materials are nowadays a standard choice for mechanical performance structures. In order for a composite structure to be optimized, technicians must use ever-evolving failure criteria assumptions and apply them to the critical points of a given structure. Today's design standards for such structures' analysis rely on advanced numerical methods, such as finite element modeling, for a more practical and accurate identification of those critical points. The purpose of this work is to develop a new **Continuum-Damage Model** based on the **LaRC03** failure criteria, and implement it as a user material model in finite element analysis code **ABAQUS** [1]. The proposed material model allows the simulation of the local material behaviour associated with damage occurrence, as well as the global structural response of a composite structure.

Resumo

Os materiais compósitos avançados são actualmente a escolha de eleição para o desenvolvimento e construção de estruturas mecânicas sujeitas aos mais altos requisitos de desempenho. A optimização de uma estrutura em materiais compósitos recorre à utilização de critérios de rotura baseados em considerações sujeitas a constante evolução, aplicada aos pontos críticos da estrutura em questão. Os métodos padrão de projecto para este tipo de estruturas baseiam-se em métodos numéricos avançados, tais como a modelação por elementos finitos, com o objectivo de tornar mais prática e precisa a identificação de esses pontos críticos. O trabalho apresentado implementa um **Modelo de Dano-Contínuo** baseado no critério de rotura **LaRC03**, como um modelo material definido pelo utilizador no código comercial de elementos finitos **ABAQUS** [1]. O modelo material proposto permite a simulação do comportamento local do material associado à ocorrência de dano, simultaneamente com a simulação da resposta estrutural global de uma estrutura em materiais compósitos.

Resumé

Actuellement les matériaux composites sont le choix préférentiel dans le développement et construction des structures mécaniques soumises aux conditions de la performance la plus élevée. L'optimisation d'une structure aux matériaux composites appelle à l'utilisation des critères de défaillance fondés sur des considérations assujetties à une évolution constante, appliquée aux points critiques de la structure en question. Les méthodes standard de projet pour ce type de structures sont basées sur des méthodes numériques avancées, comme la méthode d'éléments finis, avec l'objectif de rendre plus pratique et précise l'identification de ces points critiques. Le travail qui est présenté ici développe un **Modèle d'Endommagement-Continu** basé sur le critère de défaillance **LaRC03**, et envisagé comme un modèle matériel défini pour l'utilisateur dans le code commercial d'éléments finis **ABAQUS** [1]. Le modèle matériel proposé permet la combinaison d'une analyse de comportement et de rupture pour une structure aux matériaux composites avancés dans un seul modèle d'analyse.

Agradecimentos

Inicio por expressar o meu sincero agradecimento ao Professor Doutor Pedro Manuel Ponces Rodrigues de Castro Camanho, orientador deste trabalho, pela disponibilidade e apoio incondicionais, pelos valores de rigor científico incutidos e pelo seu lado humano revelados durante a elaboração deste trabalho.

Gostaria de apresentar um agradecimento aos docentes do Departamento de Engenharia Mecânica e Gestão Industrial pela formação científica que me proporcionaram ao longo deste anos.

Agradeço aos Professores Fernando Oliveira e Miguel Figueiredo, ao Doutor Rui Oliveira, aos Senhores José Rocha Almeida, Albino Castro Dias e Rui Martins da Silva pelo apoio e camaradagem concedidos na parte experimental deste trabalho.

Aos meus colegas do Laboratório de computação Pedro Portela, Jorge Almeida, Cassilda Tavares, Pedro Martins, Carla Roque, André Roque, Marco Parente, Jorge Belinha, David Reccio e Robertt Valente, pelo excelente ambiente de trabalho, amizade e apoio proporcionado apresento os meus sinceros agradecimentos.

Apresento os meus agradecimentos ao INEGI pelo financiamento deste trabalho e por proporcionar o acesso a muitos dos meios experimentais utilizados no decurso deste trabalho, e ao DEMEGI pelo local de trabalho e meios disponibilizados.

Gostaria finalmente de apresentar o meu sincero agradecimento à Ana, aos meus Pais e irmão, bem como aos meus amigos mais próximos por todo o apoio que me proporcionaram em situações difíceis e sem o qual a elaboração deste trabalho não teria sido possível.

À minha família e amigos,

"He who does not expect the unexpected will not detect it."

460 BC - 370 BC, Democritus

Contents

Abstract	i
Resumo	iii
Resumé	v
Agradecimentos	vii
1 Introduction	1
1.1 Motivation	1
1.2 Background	2
2 Material Selection and Characterization	5
2.1 Introduction	5
2.2 Ply properties	5
2.2.1 Fibre volume fraction	5
2.2.2 Tensile tests	6
2.2.3 Compression tests	13
2.2.4 Shear tests	18
2.3 Fracture toughness properties	21
2.3.1 Mode I fracture toughness	21
2.3.2 Mode II fracture toughness	26
2.4 Determination of the in-situ strengths	28
	xi

3	Continuum-Damage Model	31
3.1	Introduction	31
3.2	Failure criteria	32
3.3	Model description	36
4	Computational Model	39
4.1	Implicit model	44
4.1.1	Dynamic	46
4.1.2	Static	46
4.2	Explicit model	48
5	Experimental Tests	51
5.1	Introduction	51
5.2	Test matrix	51
5.3	Test results	53
5.4	Acoustic emission results	63
6	Finite Element Models	73
6.1	Introduction	73
6.2	Boundary conditions	74
6.3	Loading conditions	75
6.4	Element properties	76
7	Examples and Comparison with Experimental Results	79
7.1	Introduction	79
7.2	Load-deformation	80
7.3	Damage evolution	86
7.4	Ultimate failure load	88
8	Conclusions	91
9	Bibliography	93

List of Symbols

α_0		Angle of the fracture plane
β		Shear response factor
δ_{LVDT}	...	Average of the displacements measured by the two LVDT used in the test rig
ε_{1T}		Longitudinal tensile failure strain
ε_{2T}		Transverse tensile failure strain
ε_T^L		Laminate tensile failure strain
$\Delta\varepsilon$		Strain difference
$\Delta\varepsilon_T$		Strain difference in the transverse direction
$\Delta\varepsilon_L$		Strain difference in the longitudinal direction
ε_{xx}		Longitudinal strain
ε_{yy}		Transverse strain
${}^t\varepsilon$		Strain tensor for the current iteration
${}^{t-1}\varepsilon$		Strain tensor from the previous iteration
η^L		Coefficient of longitudinal influence
η^T		Coefficient of transverse influence
$\overline{\gamma}_{12}$		Ultimate shear strain

Φ	Failure criteria function
$\Delta\sigma$	Stress difference
$\sigma_{ij}^{(m)}$	Components of the stress tensor in a frame representing the fibre misalignment
$\bar{\sigma}^{\infty}$	Remote applied stress
$^P\sigma$	Stress tensor prediction
$^t\sigma$	Stress tensor for the current iteration
$^{t-1}\sigma$	Stress tensor from the previous iteration
σ^u	Ultimate stress
τ_{eff}	Effective shear stress
$\tau_{eff}^{(m)}$	Effective shear stress in a frame representing the fibre misalignment
ν_{12}	Major Poisson's ratio
ν_{12}^L	Laminate major Poisson's ratio
φ	Fibre misalignment angle
$\mathbf{A}$	Average area corresponding to the transverse section of a specimen
$\mathbf{A}_{total}$	Total area
$\mathbf{A}_{white}$	Area in white
d	Hole diameter
d_i	General damage variable
$\mathbf{E}_1$	Longitudinal modulus of elasticity
$\mathbf{E}_2$	Transverse modulus of elasticity

E_1^0	Undamaged longitudinal modulus of elasticity
E_2^0	Undamaged transverse modulus of elasticity
E^L	Laminate modulus of elasticity
G_{12}	In-plane shear modulus
G_{12}^0	Undamaged in-plane shear modulus
G_{IC}	Mode <i>I</i> fracture toughness for transverse crack propagation
G_{IIC}	Mode <i>II</i> fracture toughness for transverse crack propagation
K_T^∞	Stress concentration factor for infinite width
K_T	Stress concentration factor for finite width
P	Applied load
P_{max}	Maximum applied load
R_K	Finite width correction factor
S_{12}	In-plane shear strength
S_L	Tensile shear strength in the longitudinal direction
S_L^{is}	In-situ tensile shear strength in the longitudinal direction
S_{LT}^{UD}	Shear strength measured in an unidirectional test specimen
S_T	Tensile shear strength in the transverse direction
t	Thickness
t^* ..	Ply thickness corresponding to the transition between thin and thick plies
V_f	Fibre volume fraction
w	Width

w/t	Width to hole-diameter ratio
$\mathbf{X}_C^L$	Laminate compressive strength
$\mathbf{X}_T^L$	Laminate tensile strength
$\mathbf{X}_C$	Ply longitudinal compressive strength
$\mathbf{X}_T$	Ply longitudinal tensile strength
$\mathbf{Y}_C$	Ply transverse compressive strength
$\mathbf{Y}_T^{UD}$..	Tensile transverse strength measured in an unidirectional test specimen
$\mathbf{Y}_T^{is}$	In-situ ply transverse tensile strength
$\mathbf{Y}_T$	Ply transverse tensile strength

List of Abbreviations

AE		Acoustic Emission
ASTM		American Society for Testing and Materials
CFRP		Carbon Fibre Reinforced Plastics
CV		Coefficient of Variation
DCB		Double Cantilever Beam
DCB-#		Double Cantilever Beam specimen number #
DOF		Degree of Freedom
FE		Finite Element
IC		Interval of Confidence
LC-#		Laminate Compression specimen number #
LVDT		Linear Variable Displacement Transducer
OHT#-#		Open-Hole Tensile # specimen #
PC0-#		Longitudinal Compression specimen #
PC90-#		Transverse Compression specimen #
PSH-#		Shear test specimen #
PT0-#		Longitudinal Tensile specimen #

PT90-#	Transverse Tensile specimen #
PZT	Lead Zirconate Titanate
SDV	Solution Dependent Variable
STDV	Standard Deviation
TN2	Technical Note 2
TN4	Technical Note 4
UMAT	Implicit coded User Material
VUMAT	Explicit coded User Material
4-ENF	Four Point Bending End Notched Flexure specimen
4-ENF-#	Four Point Bending End Notched Flexure specimen #

List of Figures

1-1	Mesoscale constituents illustration.	3
2-1	Image processing to measure fibre volume fraction.	6
2-2	Specimen geometry type A, for the 0° tensile test.	7
2-3	Specimen geometry type B, for the 90° tensile test.	7
2-4	Stress-strain relation for the 0° specimens loaded in tension. . . .	9
2-5	Stress-strain relation for the 90° specimens loaded in tension. . . .	12
2-6	Specimen geometry type C, for the 0° compression test.	13
2-7	Specimen geometry type C, for the 90° compression test.	14
2-8	Stress-strain relation for the 0° specimens loaded in compression (absolute values).	15
2-9	Stress-strain relation for the 90° specimens loaded in compression (absolute values).	17
2-10	Specimen geometry type D, for the shear test specimen.	18
2-11	Shear stress-shear strain relation.	20
2-12	Geometry of the DCB test specimen.	21
2-13	Experimental setup for the DCB test specimen.	22
2-14	Determination of Δ for the corrected modified beam theory. . . .	23
2-15	Determination of Δ for the corrected modified beam theory. . . .	23
2-16	Crack resistance curves for the DCB test specimens.	24
2-17	Fibre bridging in a DCB test specimen.	24
2-18	Geometry of the 4-ENF test specimen.	26
2-19	Crack resistance curves for the 4-ENF test specimens.	27

3-1	Coordinate systems convention.	36
3-2	Degradation of the material elastic properties.	38
4-1	Algorithm of the UMAT user subroutine for ABAQUS standard. .	40
4-2	Algorithm of the VUMAT user subroutine for ABAQUS explicit. .	41
4-3	Flowchart of the UMAT user subroutine for ABAQUS standard. .	42
4-4	Flowchart of the VUMAT user subroutine for ABAQUS explicit. .	43
5-1	X-ray results.	52
5-2	X-ray results.	53
5-3	Load-cross head displacement for the specimens OHT1.	53
5-4	Load-cross head displacement for the specimens OHT2.	54
5-5	Load-cross head displacement for the specimens OHT3.	54
5-6	Load-cross head displacement for the specimens OHT4.	55
5-7	Load-cross head displacement for the specimens OHT5.	55
5-8	Load-cross head displacement for the specimens OHT6.	56
5-9	Load-cross head displacement for the specimens OHT7.	56
5-10	Load-cross head displacement for the specimens OHT8.	57
5-11	Load-cross head displacement for the specimens OHT9.	57
5-12	Net-section failure in an open-hole specimen.	62
5-13	Specimen OHT8 instrumented.	63
5-14	Load-AE relation for the specimen OHT1.	65
5-15	Load-AE relation for the specimen OHT2.	65
5-16	Load-AE relation for the specimen OHT3.	66
5-17	Load-AE relation for the specimen OHT4.	66
5-18	Load-AE relation for the specimen OHT5.	67
5-19	Load-AE relation for the specimen OHT6.	67
5-20	Load-AE relation for the specimen OHT7.	68
5-21	Load-AE relation for the specimen OHT8.	68
5-22	Load-AE relation for the specimen OHT9.	69
5-23	Different phases of AE in the open-hole test specimens.	70

6-1	Case study illustration.	73
6-2	Modeled geometry illustration.	74
6-3	Boundary conditions detail in the surrounding hole area.	75
7-1	General coordinates-system for the strain gauges position.	80
7-2	Load-deformation curve for the specimen OHT1.	81
7-3	Load-deformation curve for the specimen OHT2.	82
7-4	Load-deformation curve for the specimen OHT3.	82
7-5	Load-deformation curve for the specimen OHT4.	83
7-6	Load-deformation curve for the specimen OHT5.	83
7-7	Load-deformation curve for the specimen OHT6.	84
7-8	Load-deformation curve for the specimen OHT7.	84
7-9	Load-deformation curve for the specimen OHT8.	85
7-10	Load-deformation curve for the specimen OHT9.	85
7-11	Illustration of Phase A in acoustic emission events: no damage. . .	87
7-12	Illustration of Phase B in acoustic emission events: matrix cracking.	87
7-13	Illustration of Phase C in acoustic emission events: fibre cracking and ultimate failure.	87
7-14	Hole-size effect predictions with the implicit static model.	90

List of Tables

2.1	Tensile test matrix.	6
2.2	Tensile test matrix (cont.).	7
2.3	Results of the longitudinal tensile test- specimens with tapered end tabs.	10
2.4	Results of the longitudinal tensile test- specimens with straight end tabs.	11
2.5	Results of the transverse tensile test.	12
2.6	Compression test matrix.	13
2.7	Compression test matrix (cont.).	13
2.8	Results of the longitudinal compression tests.	16
2.9	Results of the transverse compression tests (absolute values). . . .	17
2.10	Shear test matrix.	18
2.11	Shear test matrix (cont.).	18
2.12	Results of the shear tests.	19
2.13	Results of the DCB tests	25
2.14	Results of the 4-ENF tests.	27
2.15	In-situ strengths of thin plies (MPa).	30
3.1	Ply elastic properties degradation scheme.	37
5.1	Open-hole tensile test matrix.	52
5.2	Results of open-hole tensile test: specimen OHT1.	58
5.3	Results of open-hole tensile test: specimen OHT2.	58
5.4	Results of open-hole tensile test: specimen OHT3.	59

5.5	Results of open-hole tensile test: specimen OHT4.	59
5.6	Results of open-hole tensile test: specimen OHT5.	60
5.7	Results of open-hole tensile test: specimen OHT6.	60
5.8	Results of open-hole tensile test: specimen OHT7.	61
5.9	Results of open-hole tensile test: specimen OHT8.	61
5.10	Results of open-hole tensile test: specimen OHT9.	62
5.11	Damage onset loads for the open-hole test specimens (values in kN).	71
6.1	DOF constraint scheme for the selected node sets.	75
6.2	Required material properties.	77
7.1	Strain gauges position.	81
7.2	Ultimate failure loads measured and predicted.	89
7.3	Ultimate failure loads prediction error.	89
7.4	Hole-size effect predictions with the implicit static model.	90

Chapter 1

Introduction

1.1 Motivation

Advanced composite materials are commonly used through the widest industry range ever. Due to their interesting specific properties these materials receive great attention from the aerospace industry in particular. The complex behaviour of composite materials leads to extensive testing campaigns to validate the design of the structures that use them; these campaigns are invariably expensive.

This poses an important industrial challenge which is to replace part of the testing campaigns with computer models that can accurately predict the behaviour and failure of such structures. Using a continuum-damage model as proposed in this work, applied to a finite element analysis that simulates either the whole structure or its critical points, significantly reduces the cost of a testing campaign.

This work focuses on the validation of the proposed continuum-damage model by comparing the finite element models' predictions with experimental results on open-hole test specimens. The prediction of the material behaviour, damage initiation, evolution and final failure in specimens containing stress concentration factors such as holes, is one of the most interesting applications for the proposed model.

1.2 Background

Damage in advanced composite materials can be seen and treated in different scales. As the observer details the approach on the material, the complexity of its treatment rises to a point where it cannot be done in real time with the computational means at our disposal in the present day. The currently accepted scales for analysis of composite material behaviour are, the:

- macroscale
- mesoscale
- microscale

The macroscale approach applies homogenization to the material. The fact that the material is composed of reinforcement fibres and an agglomerating matrix is disregarded, and it is only assumed that the material presents anisotropic elastic behaviour. At this scale, micro-defects of the material are not accounted for. The macroscale deals only with problems such as macroscopic cracks, by applying the fracture mechanics concepts, notches and large perforations [2], [3]. An analysis done at this scale has the advantage of being simple to deal with, but it also tends to present results that are a poor approach to reality due to the obvious lack of detail in the analysis. This approach often results in overdesigned structures that could be further optimized given a deeper knowledge of the material behaviour.

The mesoscale as proposed by Ladevèze [4] is the next step in detailing the material behaviour, and is applicable to laminate composite materials with different or equally angled plies. At this scale it is considered that the material is composed by two constituents, the layer and the interface as shown in Figure 1-1. The layer is a continuous three-dimensional constituent and is considered to be inelastic, damageable, homogenous and anisotropic. The interface is a two-dimensional surface that ensures stress transmission between the layers and is also considered as being damageable and therefore able to produce decohesion between two adjacent plies.

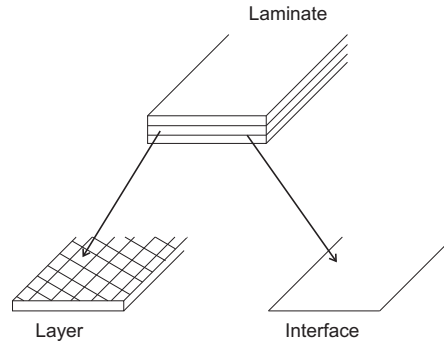


Figure 1-1: Mesoscale constituents illustration.

At the microscale level all the constituents of the composite material are individually considered and their interaction is modeled. These constituents are the reinforcement fibres, the agglomerating matrix and the fibre/matrix cohesive interface. The microscale is defined as the length within which the solid is heterogenous and piecewise continuous. Treating the material at this level will result in very accurate model predictions, but it will be impossible to compute the desired results in real time due to complexity of the material behaviour.

The mesoscale is perhaps the best compromise of accuracy and cost efficiency. An extensive amount of research papers on this subject has been published, primarily authored by Pierre Ladevèze [5]-[11] and Oliver Allix with several co-authors. In their research activities there has been an attempt to link the mesoscale approach with the micromechanics principles. This linkage tries to provide the mesoscale modeling approach with a stronger mechanical basis.

The objectives of this work are: to develop a new damage model using a physically-based failure criteria, LaRC03, to predict the onset of damage and structural collapse of composite materials; to implement the damage model in a non-linear Finite-Element (FE) code, ABAQUS [1]; to validate the model by comparing the predictions with experimental data.

In order to achieve these objectives this thesis is structured as follows: the selection and mechanical characterization of the material is presented in Chapter 2; the theoretical description of the proposed *Continuum-Damage Model* is

described in Chapter 3; the computational implementation of the damage model is presented in Chapter 4; Chapter 5 presents the experimental tests procedures and results; the FE models construction and specification is presented in Chapter 6; the comparison of results with experimental data is done in Chapter 7; and Chapter 8 details the conclusions of the presented work.

Chapter 2

Material Selection and Characterization

2.1 Introduction

The Hexcel IM7/8552 carbon epoxy unidirectional pre-preg is a space-qualified material often used in advanced structures and was the material chosen for this work. A test campaign for material characterization was planned and executed with the purpose of measuring the material properties used in the continuum-damage model.

2.2 Ply properties

2.2.1 Fibre volume fraction

The measurement of the fibre volume fraction, V_f , is a useful technique to assess the quality of the manufacturing process. It was initially attempted to measure the fibre volume fraction by burning the epoxy resin according to the ASTM standard D3171 [12]. However, the technique was not successful because part of the fibres also burned during the test.

Therefore, the fibre volume fraction was measured using image processing

techniques. Sixteen digital micrographs of the cross-section of the specimens were taken using a digital camera linked to an optical microscope. The images were afterwards modified to pure black and white using MatLab [13], and the fibre volume fraction was calculated using MatLab toolboxes as the ratio between the area in white, corresponding to the fibres, and the total area of the image, as shown in Figure 2-1.

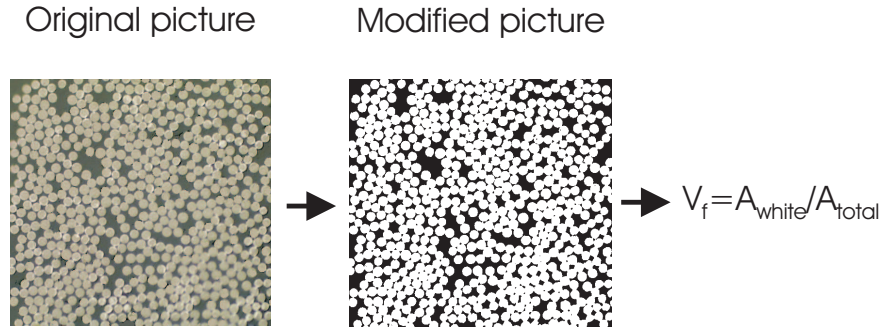


Figure 2-1: Image processing to measure fibre volume fraction.

The average fibre volume fraction measured using the process outlined above was 59.1%. The nominal fibre volume fraction provided by the manufacturer is 57.7%.

2.2.2 Tensile tests

The purpose of the tensile tests is to measure the longitudinal and transverse elastic properties and strengths of the ply. The tests were performed according to the test matrices shown in Tables 2.1 and 2.2, following the ASTM D-3039 [14] standard. The geometry of the test specimens is shown in Figures 2-2 and 2-3. The tests were performed in an Instron4200-A electro-mechanic test machine.

Table 2.1: Tensile test matrix.

Test Type	Standard	Lay-up	Objective	Geometry
Tensile	ASTM D 3039	$(0^\circ)_8$	E_1, X_T, v_{12} and ε_{1T}	A
Tensile	ASTM D 3039	$(90^\circ)_{16}$	E_2, Y_T, v_{12} and ε_{2T}	B

Table 2.2: Tensile test matrix (cont.).

Geometry	# of specimens	Instrumentation	Type of control	Speed
A	5	Strain gauges; load cell	Displacement	2 mm/min
B	5	Strain gauges; load cell	Displacement	1 mm/min

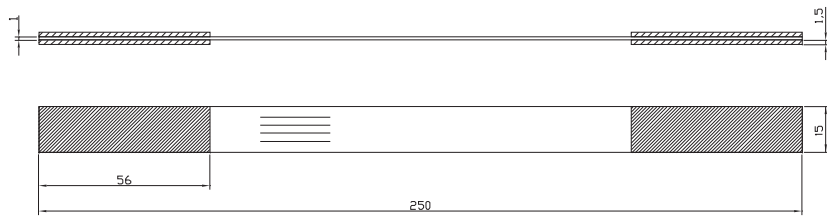


Figure 2-2: Specimen geometry type A, for the 0° tensile test.

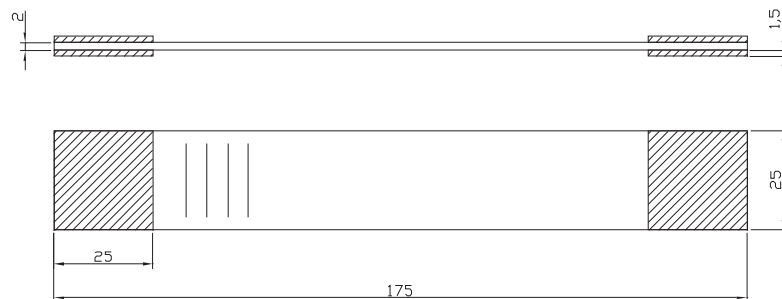


Figure 2-3: Specimen geometry type B, for the 90° tensile test.

Longitudinal tests

The longitudinal tensile test measures the following material properties:

- Longitudinal modulus of elasticity - E_1 (Pa):

$$E_1 = \frac{\Delta\sigma}{\Delta\varepsilon}. \quad (2.1)$$

where $\Delta\sigma$ (Pa) is the stress difference measured at 1000 and 3000 $\mu\varepsilon$ on the stress-strain diagram. $\Delta\varepsilon$ (m/m) is the strain difference, whose nominal value is 2000 $\mu\varepsilon$.

- Longitudinal tensile strength - X_T (Pa):

$$X_T = \frac{P_{\max}}{A} \quad (2.2)$$

where $P_{\max}$ (N) is the maximum load before fracture, and A (m²) is the average area corresponding to the transverse section of the specimen.

- Major Poisson ratio - ν_{12} , which is calculated using the following expression:

$$\nu_{12} = -\frac{\Delta\varepsilon_T}{\Delta\varepsilon_L} \quad (2.3)$$

where $\Delta\varepsilon_T$ and $\Delta\varepsilon_L$ are the strain differences in the transverse and longitudinal directions, respectively.

The 0° tensile tests were performed in specimens with both tapered and straight end tabs. The elastic properties were obtained using the tapered specimens only. Although not required by the standard, strain gages were bonded on the two surfaces of the specimens to assess the existence of bending during the tests.

Figure 2-4 shows the stress-strain curves obtained in the specimens with tapered end tabs. The strain is the average strain measured by the back-to-back strain gages.

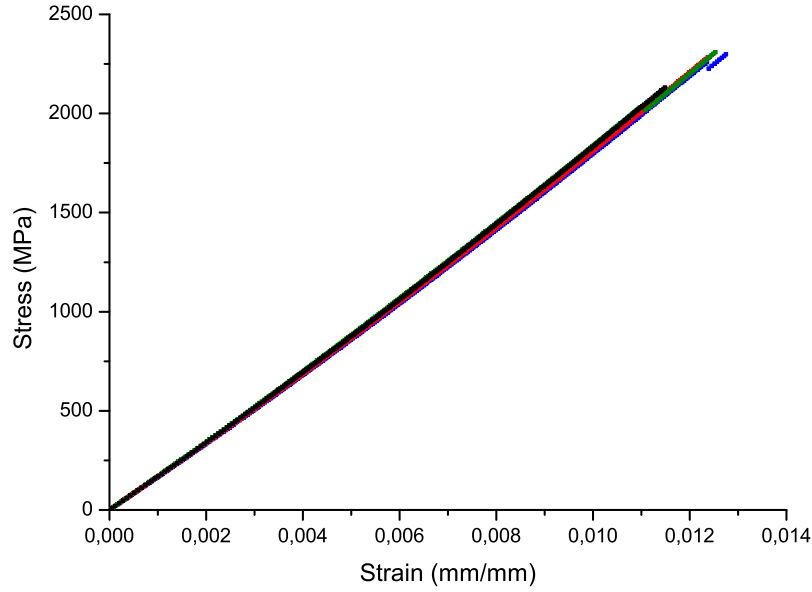


Figure 2-4: Stress-strain relation for the 0° specimens loaded in tension.

The statistical measures used to characterize the experimental data are:

Average ($\bar{x}$):

$$\bar{x} = \frac{1}{n} \sum_{i=1}^n x_i \quad (2.4)$$

where n is the number of specimens tested, and x_i corresponds to the value measured in the test.

Standard deviation, STDV, and coefficient of variation, CV, describing how spread out, or varied, the observations are:

$$\text{STDV} = S_{n-1} = \sqrt{\frac{1}{n-1} \sum_{i=1}^n (x_i^2 - n\bar{x}^2)} \quad (2.5)$$

$$\text{CV} = 100 \frac{S_{n-1}}{\bar{x}} \quad (2.6)$$

Confidence interval at 95%, representing an estimate of the real mean value of the variable in the population:

$$\text{IC} = \bar{x} \pm t_{n-1(2.5\%)} \frac{S_{n-1}}{\sqrt{n}} \quad (2.7)$$

where $t_{n-1(2.5\%)}$ is the value of the t distribution for a symmetrical 95% confidence interval.

Tables 2.3 and 2.4 show the results obtained in the 0° tensile tests.

Table 2.3: Results of the longitudinal tensile test- specimens with tapered end tabs.

Spec. Ref.	w (mm)	t (mm)	E_1 (GPa)	ν_{12}	X_T (MPa)
PT01T	15.01	0.98	138.84	0.31	2426.95
PT02T	15.00	0.98	169.83	0.33	2298.23
PT03T	15.00	0.98	170.57	0.29	2283.27
PT04T	15.00	0.99	174.17	0.34	2308.42
PT05T	15.00	0.99	173.68	0.31	2131.18
Average	15.00	0.98	171.42	0.32	2289.61
STDV	-	-	2.38	0.02	105.39
CV (%)	-	-	1.39	6.18	4.60
IC	-	-	± 2.95	± 0.02	± 130.84

Table 2.4: Results of the longitudinal tensile test- specimens with straight end tabs.

Spec. Ref.	w (mm)	t (mm)	X _T (MPa)
PT01S	15.00	1.00	2307.20
PT02S	14.99	1.00	2409.87
PT03S	15.00	0.99	2245.12
PT04S	15.00	1.01	2229.97
PT05S	15.01	1.01	2594.44
Average	15.00	1.00	2357.32
STDV	-	-	150.26
CV (%)	-	-	6.37
IC	-	-	±186.54

Transverse tests

The transverse tensile test measures the following material properties:

- Transverse modulus of elasticity - E_2 (Pa):

$$E_2 = \frac{\Delta\sigma}{\Delta\varepsilon}. \quad (2.8)$$

where $\Delta\sigma$ (Pa) is the stress difference corresponding to the 1000 and 3000 $\mu\varepsilon$ points on the stress-strain diagram. $\Delta\varepsilon$ (m/m) is the strain difference, whose nominal value is 2000 $\mu\varepsilon$.

- Transverse tensile strength - Y_T (Pa):

$$Y_T = \frac{P_{\max}}{A} \quad (2.9)$$

where $P_{\max}$ (N) is the maximum load before fracture and A (m²) is the medium area corresponding to the transverse section of the specimen.

The results of the tests performed are shown in Figure 2-5.

Table 2.5 shows the results obtained in the 90° tensile tests.

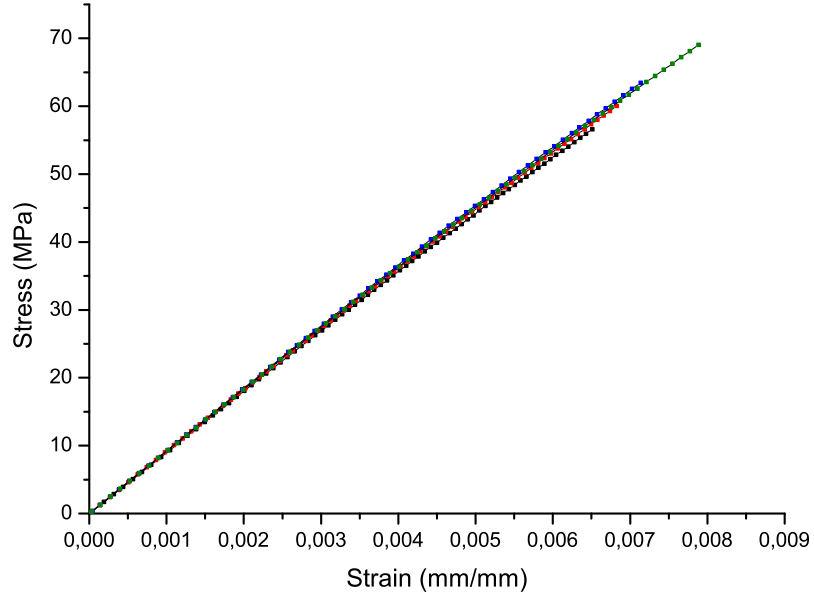


Figure 2-5: Stress-strain relation for the 90° specimens loaded in tension.

Table 2.5: Results of the transverse tensile test.

Spec. Ref.	w (mm)	t (mm)	E_2 (GPa)	Y_T (MPa)
PT90-1	24.72	1.99	9.02	-
PT90-2	24.57	1.99	8.98	56.61
PT90-3	24.67	1.99	9.04	60.05
PT90-4	24.74	1.99	9.21	63.45
PT90-5	24.75	1.98	9.13	69.03
Average	24.69	1.99	9.08	62.29
STDV	-	-	0.09	5.29
CV (%)	-	-	1.03	8.50
IC	-	-	± 0.12	± 8.42

2.2.3 Compression tests

The purpose of the compression tests is to measure the strengths of the ply under compressive loading, used in ply-based failure criteria.

The tests were performed according to the test matrices shown in Tables 2.6 and 2.7. Strain gages were bonded on the two surfaces of the laminates to assess the existence of column buckling during the tests. The tests were performed following the ASTM standard D-3410 [15].

Table 2.6: Compression test matrix.

Test Type	Standard	Lay-up	Objective	Geometry
Compression	ASTM D 3410	$(0^\circ)_{16}$	$X_C, \varepsilon_{1C}, \nu_{12}$	C
Compression	ASTM D 3410	$(90^\circ)_{16}$	$Y_C, \varepsilon_{2C}, \nu_{12}$	C

Table 2.7: Compression test matrix (cont.).

Geometry	# of specimens	Instrumentation	Type of control	Speed
C	5	Strain gauges; load cell	Displacement	1.125 mm/min
C	5	Strain gauges; load cell	Displacement	1.125 mm/min

The geometry of the test specimens is shown in Figures 2-6 and 2-7.

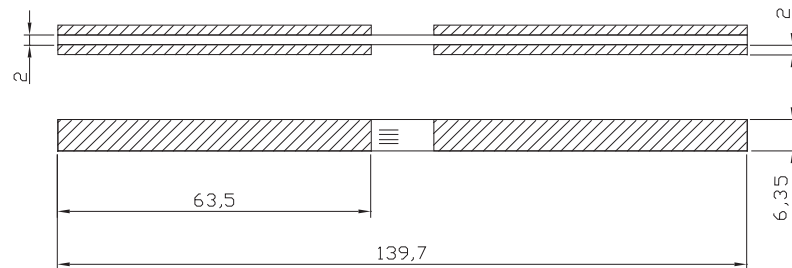


Figure 2-6: Specimen geometry type C, for the 0° compression test.

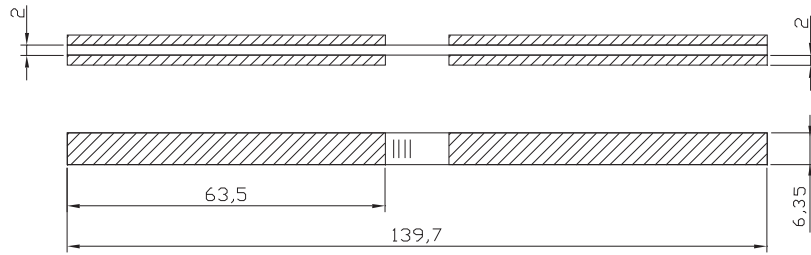


Figure 2-7: Specimen geometry type C, for the 90° compression test.

Longitudinal test

The longitudinal compression tests allow the measurement of the following mechanical properties:

- Longitudinal compressive strength - X_C (Pa), which is obtained from the 0° compression test and determined using the following expression:

$$X_C = \frac{P_{\max}}{A} \quad (2.10)$$

where $P_{\max}$ (N) is the maximum load before fracture, and A (m²) is the average area corresponding to the transverse section of the specimens.

The results of the tests performed are shown in Figure 2-8.

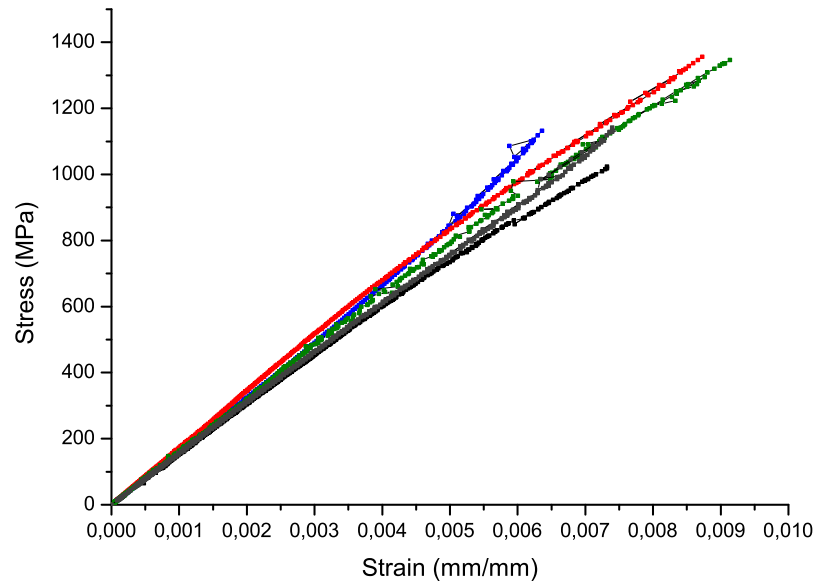


Figure 2-8: Stress-strain relation for the 0° specimens loaded in compression (absolute values).

Table 2.8 shows the results obtained in the 0° compression tests.

Table 2.8: Results of the longitudinal compression tests.

Spec. Ref.	w (mm)	t (mm)	X _C (MPa)
PC0-1	6.05	2	1132.23
PC0-2	6.05	2	1023.47
PC0-4	6.05	2	1356.03
PC0-5	6.05	2	1346.45
PC0-7	6.05	2	1142.15
Average	6.05	2	1200.07
STDV	-	-	145.68
CV (%)	-	-	12.14
IC	-	-	±180.86

Transverse test

The transverse compression tests allow the measurement of the following mechanical properties:

- Transverse compressive strength - Y_C (Pa), which is obtained from the 90° compression test and determined using the following equation:

$$Y_C = \frac{P_{\max}}{A} \quad (2.11)$$

where $P_{\max}$ (N) is the maximum load before fracture, and A (m²) is the average area corresponding to the transverse section of the specimens.

The results of the tests performed are shown in Figure 2-9.

Table 2.8 shows the results obtained in the 90° compression tests.

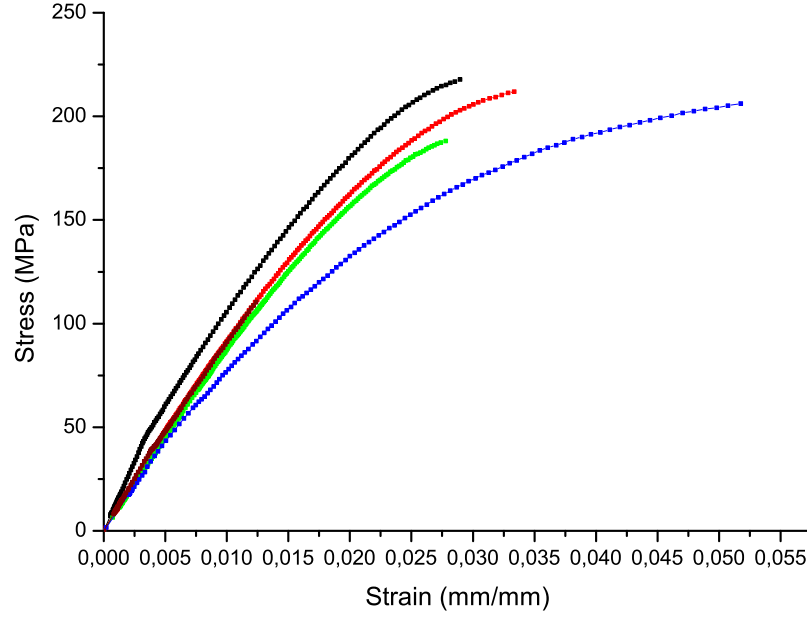


Figure 2-9: Stress-strain relation for the 90° specimens loaded in compression (absolute values).

Table 2.9: Results of the transverse compression tests (absolute values).

Spec. Ref.	w (mm)	t (mm)	Y_C (MPa)
PC90-1	6.02	2	169.4
PC90-2	6.02	2	217.4
PC90-3	6.02	2	211.6
PC90-4	6.02	2	188.1
PC90-5	6.02	2	212.6
Average	6.02	2	199.81
STDV	-	-	20.48
CV (%)	-	-	10.25
IC	-	-	± 25.43

2.2.4 Shear tests

The shear tests were performed according to the test matrix shown in Tables 2.10 and 2.11 and followed ASTM standard D-3518 [16].

Table 2.10: Shear test matrix.

Test Type	Standard	Lay-up	Objective	Geometry
Shear	ASTM D 3518	$(45^\circ/-45^\circ)_{4s}$	G_{12} , S_{12} , $\bar{\gamma}_{12}$	D

Table 2.11: Shear test matrix (cont.).

Geometry	# of specimens	Instrumentation	Type of control	Speed
D	5	Strain gauges; load cell	Displacement	1 mm/min

The geometry of the shear test specimen is shown in Figure 2-10.

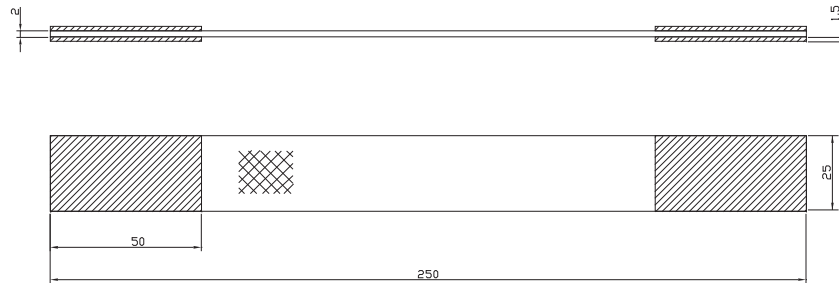


Figure 2-10: Specimen geometry type D, for the shear test specimen.

The shear tests allow the measurement of the following mechanical properties:

- In-plane shear modulus - G_{12} (Pa), obtained by the following equation:

$$G_{12} = \frac{\Delta\sigma_{12}}{\Delta\gamma_{12}} \quad (2.12)$$

where $\Delta\sigma_{12}$ (Pa) corresponds to the difference in applied shear stress corresponding to the difference between the two shear strain points (nominally 0.004).

- In-plane shear strength – S_L , obtained using the following expression:

$$S_L = \frac{P_{\max}}{2A} \quad (2.13)$$

where $P_{\max}$ (N) is the maximum load before failure, and A (m²) is the average area of the transverse section of the specimen.

- Ultimate shear strain- $\bar{\gamma}_{12}$, obtained by the equation:

$$\bar{\gamma}_{12} = \varepsilon_{xx} - \varepsilon_{yy} \quad (2.14)$$

where ε_{xx} corresponds to the longitudinal normal strain at failure and ε_{yy} is the transverse normal strain at failure.

In addition, the shear test is used to calculate the shear response factor, β , required for the calculation of the in-situ shear strength [17]-[18]. The determination of the shear response factor is described in point 2.4.

The results of the tests performed are shown in Table 2.12.

Figure 2-11 shows the shear stress-engineering shear strain relation obtained in all the tests performed.

Table 2.12: Results of the shear tests.

Spec. Ref.	w (mm)	t (mm)	S_L (MPa)	G_{12} (GPa)
PSH-1	24.75	1.99	93.07	5.42
PSH-2	24.81	1.99	92.81	5.20
PSH-3	24.63	2.00	92.00	5.30
PSH-4	24.75	1.99	91.53	5.11
PSH-5	24.81	1.99	92.28	5.40
Average	24.75	1.99	92.34	5.29
STDV	-	-	0.62	0.13
CV (%)	-	-	0.67	2.53
IC	-	-	± 0.77	± 0.17

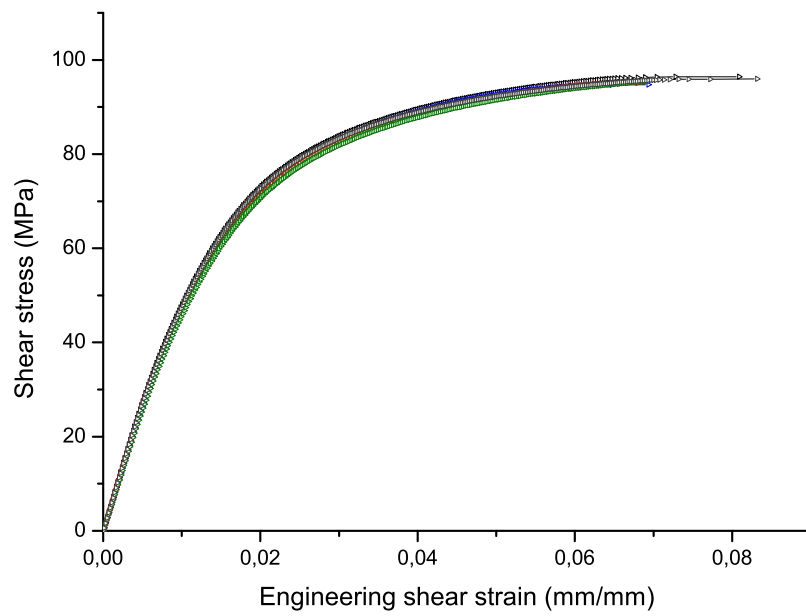


Figure 2-11: Shear stress-shear strain relation.

2.3 Fracture toughness properties

The objective of the fracture tests is to measure the components of the fracture toughness in mode I and mode II, G_{IC} and G_{IIC} respectively, of the composite laminate.

For this purpose, double cantilever beam (DCB) and 4 point bending end notched flexure (4-ENF) tests were performed to measure the mode I and mode II components of the energy release rate. The values of G_{IC} and G_{IIC} are essential for the computation of the in situ strengths required by the LaRC03/04 criteria.

2.3.1 Mode I fracture toughness

The DCB test was performed in accordance with the ASTM D5528-01 standard [19]. Four specimens with the geometry for the shown in Figure 2-12 were tested. The crack was obtained by placing a teflon insert between the composite layers prior to curing.

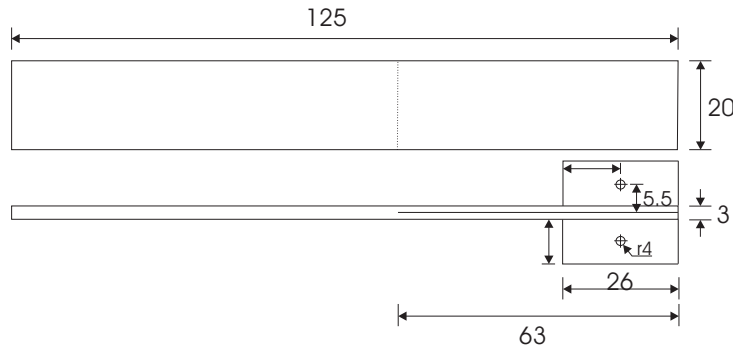


Figure 2-12: Geometry of the DCB test specimen.

Test procedure and results

The edges of the specimen were coated with typewriter correction fluid before testing. Lines separated by 5mm were marked in the specimen immediately after the crack tip. The specimens were pre-cracked, and the delamination length was measured. The DCB test was performed using a crosshead displacement of

0.5mm/min. During the test several pictures of the specimen were taken with the purpose of measuring the crack length. Figure 2-13 shows the test setup.

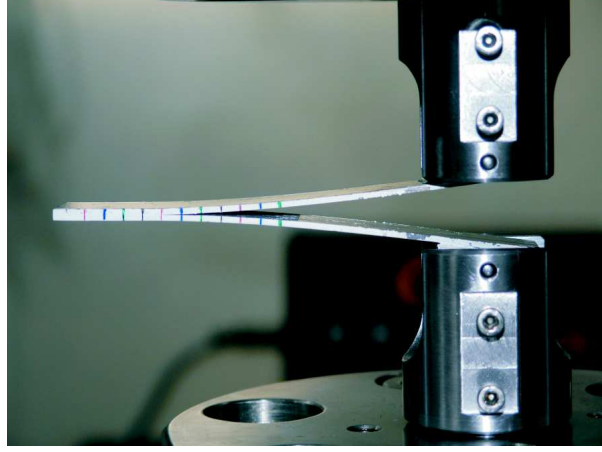


Figure 2-13: Experimental setup for the DCB test specimen.

The mode I Interlaminar fracture toughness, G_{IC} , is calculated according to the corrected modified beam theory:

$$G_{IC} = \frac{3P(\delta/N)}{2B(a + \Delta)} F$$

where:

- P corresponds to the applied load.
- δ is the displacement measured by the LVDT.
- B is the specimen width.
- a is the delamination length.
- Δ corresponds to the delamination length correction, determined experimentally by generating a least squares plot of $(C/N)^{1/3}$, as function of delamination length, Figure 2-14. (C is defined as δ/P)
- N and F are correction factors defined by:

$$F = 1 - \frac{3}{10} \left(\frac{\delta}{a} \right)^2 - \frac{3}{2} \left(\frac{\delta t}{a^2} \right)$$

$$N = 1 - \left(\frac{L'}{a}\right)^3 - \frac{9}{8} \left[1 - \left(\frac{L'}{a}\right)^2\right] \left(\frac{\delta t}{a^2}\right) - \frac{9}{35} \left(\frac{\delta}{a}\right)^2$$

t and L' are shown in Figure 2-15

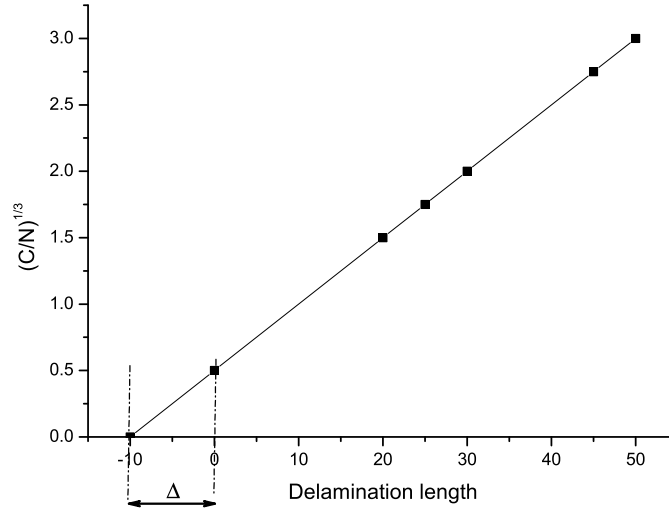


Figure 2-14: Determination of Δ for the corrected modified beam theory.

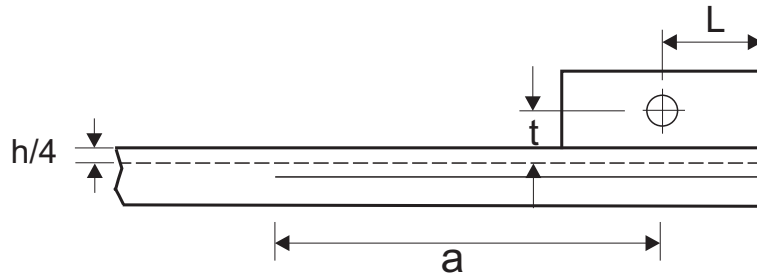


Figure 2-15: Determination of Δ for the corrected modified beam theory.

Figure 2-16 show the delamination resistance curves, obtained in the experimental tests.

Table 2.13 show the values of the mode I fracture toughness measured in the experimental tests.

The values of G_{IC} presented in Table 2.13 correspond to the onset of crack propagation. The value of G_{IC} increases with the delamination length, as shown

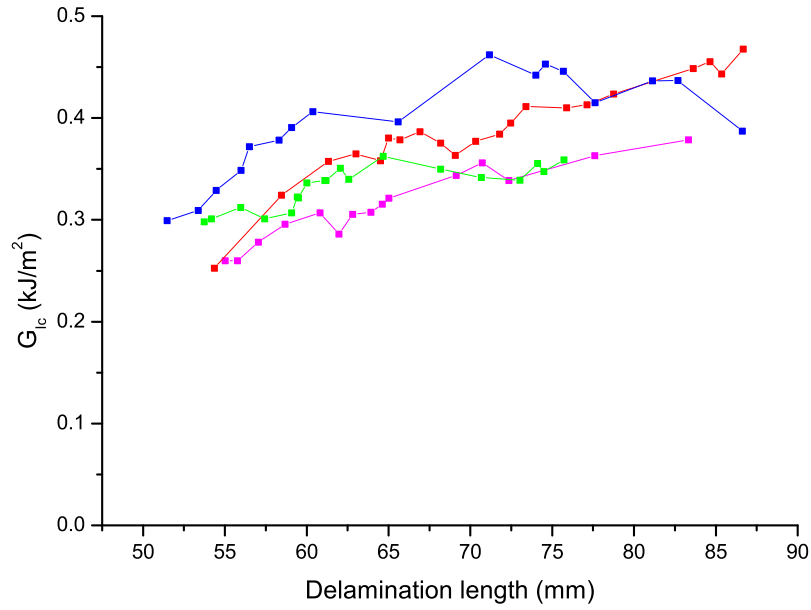


Figure 2-16: Crack resistance curves for the DCB test specimens.

in Figure 2-16. This effect is a result of the fibre bridging that is shown in detail in Figure 2-17. It is considered that fibre bridging is an artifact of the DCB test specimen and does not occur in multidirectional laminates. Therefore, the initiation value was used for G_{IC} .



Figure 2-17: Fibre bridging in a DCB test specimen.

Table 2.13: Results of the DCB tests

Spec. Ref.	G_{IC} (kJ/m ²)
DCB-1	0.2600
DCB-3	0.2525
DCB-4	0.2991
DCB-5	0.2980
Average	0.2774
STDV	0.0246
CV (%)	0.88
IC	0.047

2.3.2 Mode II fracture toughness

There is no standard test method to measure the mode II component of the energy release rate. The 4-ENF test was the selected because it generally leads to stable crack propagation.

Four specimens were tested. The specimen geometry used for the 4-ENF test is presented in Figure 2-18.

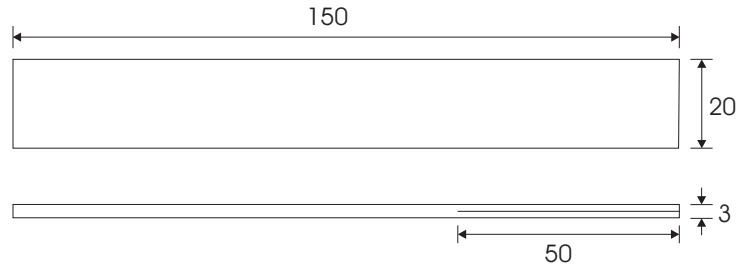


Figure 2-18: Geometry of the 4-ENF test specimen.

Test procedure and results

The edges of the specimen were coated with typewriter correction fluid before testing. Lines separated by 5mm were marked in the specimen immediately after the crack tip.

The specimens were pre-cracked in mode I, and the resulting crack length was measured. The 4-ENF test was performed using a crosshead displacement of 0.2mm/min. During the test several pictures of the specimen were taken with the purpose of measuring the length of the crack.

The mode II Interlaminar fracture toughness, G_{IIC} , is calculated as:

$$G_{IIC} = \frac{mP^2}{2B}$$

where:

- m corresponds to the slope of the linear relationship between compliance C and delamination length.
- P corresponds to the applied load.

- B is the specimen width.

Figure 2-19 show the crack resistance curve obtained in the tests.

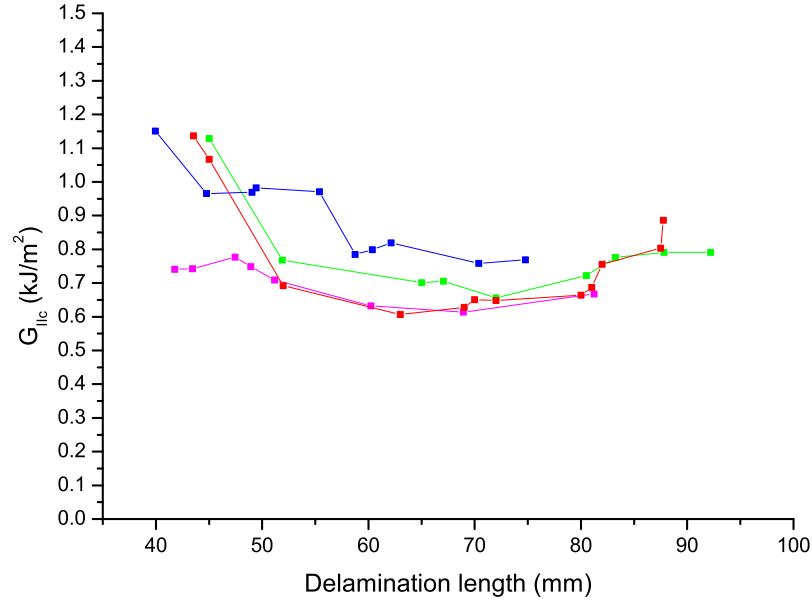


Figure 2-19: Crack resistance curves for the 4-ENF test specimens.

The test results obtained in the 4-ENF test are shown in Table 2.14.

Table 2.14: Results of the 4-ENF tests.

Spec. Ref.	G_{IIc} (kJ/m ²)
4-ENF-1	0.7039
4-ENF-2	0.7823
4-ENF-3	0.8969
4-ENF-4	0.7687
Average	0.7879
STDV	0.0803
CV (%)	10.1927
IC	0.1530

2.4 Determination of the in-situ strengths

The experimental data reported in this Chapter is required to calculate the 'in-situ' strengths of the plies. The 'in-situ' tensile transverse and in-plane shear strengths correspond to the real strength of a ply when it is embedded in a multidirectional laminate. These strengths are higher than the ones obtained in unidirectional laminates, such as the ones used in the ASTM standards. In-situ strengths will be used in the continuum-damage model applied to the finite element simulations.

The 'in-situ' transverse tensile strength is calculated as [17]-[18],[20]:

$$\text{For a thin embedded ply: } Y_T^{is} = \sqrt{\frac{8G_{IC}}{\pi t \Lambda_{22}^o}} \quad (2.15)$$

$$\text{For a thin outer ply: } Y_T^{is} = 1.79 \sqrt{\frac{G_{IC}}{\pi t \Lambda_{22}^o}} \quad (2.16)$$

$$\text{For a thick ply: } Y_T^{is} = 1.12\sqrt{2}Y_T^{UD} \quad (2.17)$$

where Y_T^{UD} is the tensile transverse strength measured in an unidirectional test specimen, t is the ply thickness, G_{IC} is the Mode I fracture toughness for transverse crack propagation, and Λ_{22}^o is defined as:

$$\Lambda_{22}^o = 2 \left(\frac{1}{E_2} - \frac{\nu_{21}^2}{E_1} \right) = 2.2 \times 10^{-4} \text{MPa}^{-1} \quad (2.18)$$

using $\nu_{21} = \nu_{12} \frac{E_2}{E_1} = 0.017$.

The ply thickness corresponding to the transition between thin and thick plies, t^* , is obtained from equations (2.15) and (2.17) as:

$$t^* = \frac{8G_{IC}}{\pi \Lambda_{22}^o (1.12\sqrt{2}Y_T^{UD})^2} \quad (2.19)$$

The in-situ tensile shear strengths are obtained as [17]:

$$S_L^{is} = \sqrt{\frac{(1 + \beta\phi G_{12}^2)^{1/2} - 1}{3\beta G_{12}}} \quad (2.20)$$

where β is the shear response factor, and the parameter ϕ is defined according to the configuration of the ply:

$$\begin{aligned} \text{For a thick ply:} \quad \phi &= \frac{12 (S_L^{\text{UD}})^2}{G_{12}} + 18\beta (S_L^{\text{UD}})^4 \\ \text{For a thin ply:} \quad \phi &= \frac{48G_{\text{IIC}}}{\pi t} \\ \text{For an outer ply:} \quad \phi &= \frac{24G_{\text{IIC}}}{\pi t} \end{aligned} \quad (2.21)$$

where S_L^{UD} is the shear strength measured in an unidirectional test specimen, and G_{IIC} is the Mode II fracture toughness for transverse crack propagation.

The shear response factor is required to calculate the in-situ shear strength, i.e., the shear strength of a ply when it is embedded in a multidirectional laminate. Tsai proposed approximating the non-linear shear response with the following polynomial [21]:

$$\gamma_{12} = \frac{1}{G_{12}}\sigma_{12} + \beta\sigma_{12}^3 \quad (2.22)$$

where β defines the non-linearity of the shear stress-shear strain relation, which is zero for a linear behavior. The shear response factor was calculated from a least-squares approximations of test data. Considering that the 5 experimental tests performed yielded similar shear stress-shear strain relations, the results obtained in the specimen PSH-2 were used to calculate β . Taking into account that for shear deformations higher than 5% the formulae proposed by the ASTM standard are no longer valid due to the finite fibre rotations, the experimental points corresponding to shear deformations higher than 5% are not used in the least-squares fit.

A Maple routine was developed for an automatic calculation of the shear re-

sponse factor. The shear response factor was calculated as $\beta = 2.98 \times 10^{-8} \text{MPa}^{-3}$.

The ply thickness corresponding to the transition between thin and thick plies, t^{**} , is obtained from equations (2.20) and (2.21) as:

$$t^{**} = \frac{8G_{\text{IIC}}G_{12}}{\pi (S_L^{\text{UD}})^2 \left[2 + 3\beta^2 G_{12} (S_L^{\text{UD}})^2 \right]^2} \quad (2.23)$$

The shear strength in the transverse direction is calculated as:

$$S_T = Y_C \cos \alpha_0 \left(\sin \alpha_0 + \frac{\cos \alpha_0}{\tan 2\alpha_0} \right) \quad (2.24)$$

For a fracture angle $\alpha_0 = 53^\circ$, the shear strength in the transverse direction is calculated from the previous equation as $S_T = 75.3 \text{MPa}$.

The in-situ strengths are presented in Table 2.15.

Table 2.15: In-situ strengths of thin plies (MPa).

Ply configuration	Y_T^{is}	S_L^{is}
Thin embedded ply	160.2	130.2
Thin outer ply	101.4	107.0
Thick ply	98.7	113.1

Chapter 3

Continuum-Damage Model

3.1 Introduction

The proposed continuum-damage model is based on three major constituents: stress analysis, damage activation functions and damage evolution laws.

The stress distribution is obtained from any standard finite element (FE) code.

The damage activation functions used correspond to the LaRC03 failure criteria described in Section 3.2. Using the LaRC03 failure criteria and the stress distribution obtained from a FE model it is possible to predict the damage initiation and evolution at the desired material points.

3.2 Failure criteria

The proposed model implements the LaRC03 [20] failure criteria.

The LaRC03 criteria treats fibre and matrix separately and is used to predict damage initiation and propagation at ply level allowing to distinguish fibre failure from matrix failure under all loading conditions.

The failure criteria is composed of six equations used to define the failure envelope as follows:

Transverse fracture

Tension The LaRC03 criterion to predict failure under transverse tension ($\sigma_{22} \geq 0$) and in-plane shear is defined as:

$$\begin{aligned} (1 - g) \frac{\sigma_{22}}{Y_T^{is}} + g \left(\frac{\sigma_{22}}{Y_T^{is}} \right)^2 + \left(\frac{\sigma_{12}}{S_L^{is}} \right)^2 - 1 &\leq 0 \\ (1 - g) \frac{\sigma_{22}^{(m)}}{Y_T^{is}} + g \left(\frac{\sigma_{22}^{(m)}}{Y_T^{is}} \right)^2 + \left(\frac{\sigma_{12}^{(m)}}{S_L^{is}} \right)^2 - 1 &\leq 0 \\ \sigma_{11} < 0, |\sigma_{11}| < X_C/2 & \end{aligned} \quad (3.1)$$

where $g = \frac{G_{Ic}}{G_{IIc}}$.

Compression The failure criteria used to predict fracture under transverse compression ($\sigma_{22} < 0$) and in-plane shear is defined as:

$$\left(\frac{\tau_{eff}^T}{S_T} \right)^2 + \left(\frac{\tau_{eff}^L}{S_L^{is}} \right)^2 - 1 \leq 0, \sigma_{11} \geq Y_C \quad (3.2)$$

$$\left(\frac{\tau_{eff}^{(m)T}}{S_T} \right)^2 + \left(\frac{\tau_{eff}^{(m)L}}{S_L^{is}} \right)^2 - 1 \leq 0, \sigma_{11} < Y_C \quad (3.3)$$

The effective shear stresses in the fracture plane are defined as:

$$\tau_{eff}^T = \langle |\tau^T| + \eta^T \sigma_n \cos \theta \rangle \quad (3.4)$$

$$\tau_{eff}^L = \langle |\tau^L| + \eta^L \sigma_n \sin \theta \rangle \quad (3.5)$$

with $\theta = \tan^{-1} \left(\frac{-|\sigma_{12}|}{\sigma_{22} \sin \alpha} \right)$. $\langle x \rangle$ is the McAuley operator defined as $\langle x \rangle := \frac{1}{2} (x + |x|)$.

The components of the stress tensor on the fracture plane are given by:

$$\begin{cases} \sigma_n = \sigma_{22} \cos^2 \alpha \\ \tau^T = -\sigma_{22} \sin \alpha \cos \alpha \\ \tau^L = \sigma_{12} \cos \alpha \end{cases} \quad (3.6)$$

The terms $\tau_{eff}^{(m)T}$ and $\tau_{eff}^{(m)L}$ are calculated from equations (3.4)-(3.5) using the relevant components of the stress tensor established in a frame representing the fibre misalignment. The fracture plane is defined by the angle α . The determination of α is performed numerically, in the failure criteria LaRC03, maximizing equation (3.2).

The coefficients of transverse and longitudinal influence, η^T and η^L respectively, can be obtained as:

$$\eta^T = \frac{-1}{\tan 2\alpha_0} \quad (3.7)$$

$$\eta^L = -\frac{S_L^{\text{is}} \cos 2\alpha_0}{Y_C \cos^2 \alpha_0} \quad (3.8)$$

where α_0 is the fracture angle under pure transverse compression ($\alpha_0 \approx 53^\circ$).

In the absence of test data the transverse shear strength can be estimated as:

$$S_T = Y_C \cos \alpha_0 \left(\sin \alpha_0 + \frac{\cos \alpha_0}{\tan 2\alpha_0} \right) \quad (3.9)$$

Longitudinal failure

Tension The failure criterion used to predict fibre fracture under longitudinal tension ($\sigma_{11} \geq 0$) is defined as:

$$\frac{\sigma_{11}}{X_T} - 1 \leq 0 \quad (3.10)$$

Compression The failure criterion used to predict fibre fracture under longitudinal compression ($\sigma_{11} < 0$) and in-plane shear (fibre kinking) is established as a function of the components of the stress tensor in a frame representing the fibre misalignment, $\sigma_{ij}^{(m)}$:

$$\begin{aligned} \sigma_{11}^{(m)} &= \sigma_{11} \cos^2 \varphi + \sigma_{22} \sin^2 \varphi + 2 |\sigma_{12}| \sin \varphi \cos \varphi \\ \sigma_{22}^{(m)} &= \sigma_{11} \sin^2 \varphi + \sigma_{22} \cos^2 \varphi - 2 |\sigma_{12}| \sin \varphi \cos \varphi \\ \sigma_{12}^{(m)} &= -\sigma_{11} \sin \varphi \cos \varphi + \sigma_{22} \sin \varphi \cos \varphi + \\ &\quad + |\sigma_{12}| (\cos^2 \varphi - \sin^2 \varphi) \end{aligned} \quad (3.11)$$

where the misalignment angle φ is defined as:

$$\varphi = \frac{|\sigma_{12}| + (G_{12} - X_C) \varphi^c}{G_{12} + \sigma_{11} - \sigma_{22}} \quad (3.12)$$

$$\varphi^c = \tan^{-1} \left[\frac{1 - \sqrt{1 - 4\varpi \left(\frac{S_L^{\text{is}}}{X_C} \right)}}{2\varpi} \right] \quad (3.13)$$

with $\varpi = \frac{S_L^{\text{is}}}{X_C} + \eta^L$.

The criteria for fibre kinking are defined as:

$$\begin{aligned}
 & \left\langle \frac{|\sigma_{12}^{(m)}| + \eta^L \sigma_{22}^{(m)}}{S_L^{\text{is}}} \right\rangle - 1 \leq 0, \quad \sigma_{22}^{(m)} < 0 \\
 & (1 - g) \frac{\sigma_{22}^{(m)}}{Y_T^{\text{is}}} + g \left(\frac{\sigma_{22}^{(m)}}{Y_T^{\text{is}}} \right)^2 + \left(\frac{\sigma_{12}^{(m)}}{S_L^{\text{is}}} \right)^2 - 1 \leq 0, \\
 & \sigma_{22}^{(m)} \geq 0, |\sigma_{11}| \geq X_C/2
 \end{aligned} \tag{3.14}$$

The LaRC03 failure criteria is completely based on mechanical and physics principles, not requiring curve fitting to experimental data.

The integration of the failure criteria in the proposed continuum-damage model is described in Section 3.3.

3.3 Model description

For a better understanding of the model, a brief illustration of the coordinate systems used is shown in Figure 3-1. The X-Y-Z system is the global coordinates system. The 1-2-3 system is the local material coordinates system defined for each ply with the 1 axis representing the fibre direction, the 2 axis representing the direction perpendicular to the fibre direction and the 3 axis representing the out-of-plane direction.

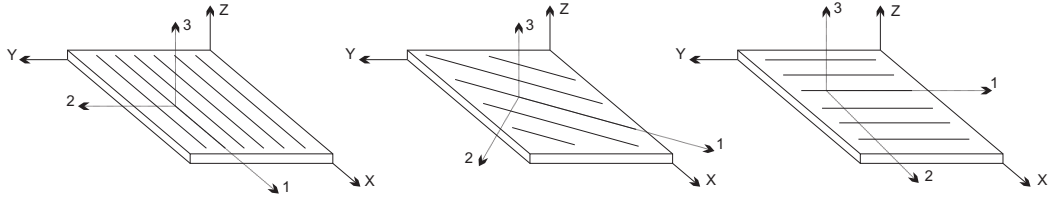


Figure 3-1: Coordinate systems convention.

The implementation of the proposed model in a non-linear finite element code provides accurate data on the stress distribution. Local stress concentration factors such as edge effects or the presence of a hole are contemplated therefore eliminating the typical limitations of analytical predictions.

Given the stress distribution, the failure criteria is applied to every material point to evaluate the damage state in each load increment.

If damage occurrence is predicted, the corresponding elastic material property is reduced as follows.

The proposed model is based on the immediate reduction of the ply elastic properties (*integration point discount method*). The ply elastic properties are reduced as a function of the type of damage predicted by the LaRC03 [20] failure criteria, as shown in Table 3.1.

Taking Φ as the failure criteria function it is possible to define the proposed model as shown in Equations (3.15) and (3.16), where d_i represents a general damage variable.

Table 3.1: Ply elastic properties degradation scheme.

Damage mechanism	E_1^d	E_2^d	E_3^d	G_{12}^d	G_{23}^d
Transverse tensile failure	E_1	$0.2E_2$	$0.2E_3$	$0.2G_{12}$	$0.2G_{23}$
Transverse compressive failure	E_1	E_2	E_3	$0.4G_{12}$	G_{23}
Longitudinal tensile failure	$0.05E_1$	E_2	E_3	G_{23}	G_{23}
Longitudinal compressive failure	$0.05E_1$	E_2	E_3	G_{23}	G_{23}

$$\Phi(\sigma_{ij}) < 1 \Rightarrow d_i = 0 \quad (3.15)$$

$$\Phi(\sigma_{ij}) \geq 1 \Rightarrow d_i \neq 0 \quad (3.16)$$

A total of three damage variables are used in the model, and are related to fibre fracture, and to matrix cracking.

Equations (3.17) to (3.19) show the application of the damage variables to the material elastic properties.

$$E_1 = (1 - d_1) E_1^0 \quad (3.17)$$

$$E_2 = (1 - d_2) E_2^0 \quad (3.18)$$

$$G_{12} = (1 - d_6) G_{12}^0 \quad (3.19)$$

Figure 3-2 illustrates the degradation scheme of the elastic properties ¹ as described in Table 3.1.

After the prediction of damage occurrence and updating the elastic material properties, the new stress distribution data is passed to the FE code for another equilibrium iteration and load increment.

Damage propagation and accumulation continues up to ultimate failure.

The non-linear finite elements analysis code provides a ply-wise stress and

¹Note that the value of the degradation variable $d=1$ is merely indicative, as the real values of these variables are given in Table 3.1.

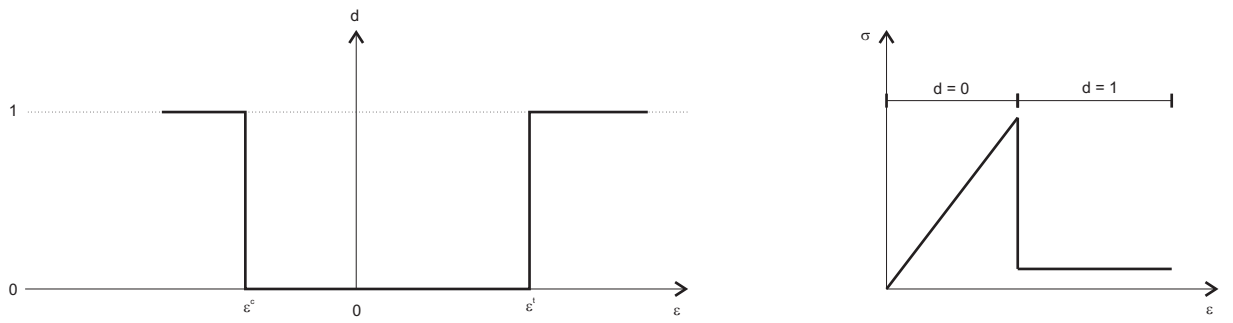


Figure 3-2: Degradation of the material elastic properties.

damage distribution computation, allowing the user to have a comprehensive view of the damage evolution process as a whole.

Chapter 4

Computational Model

Abaqus [1] non-linear finite element code was used to implement the methodology proposed. Two Fortran subroutines were developed: an Abaqus UMAT (User MATerial) to be used with ABAQUS standard (implicit), and an Abaqus VUMAT to be used with ABAQUS explicit. The algorithms corresponding to the proposed material model are shown in Figures 4-1 and 4-2.

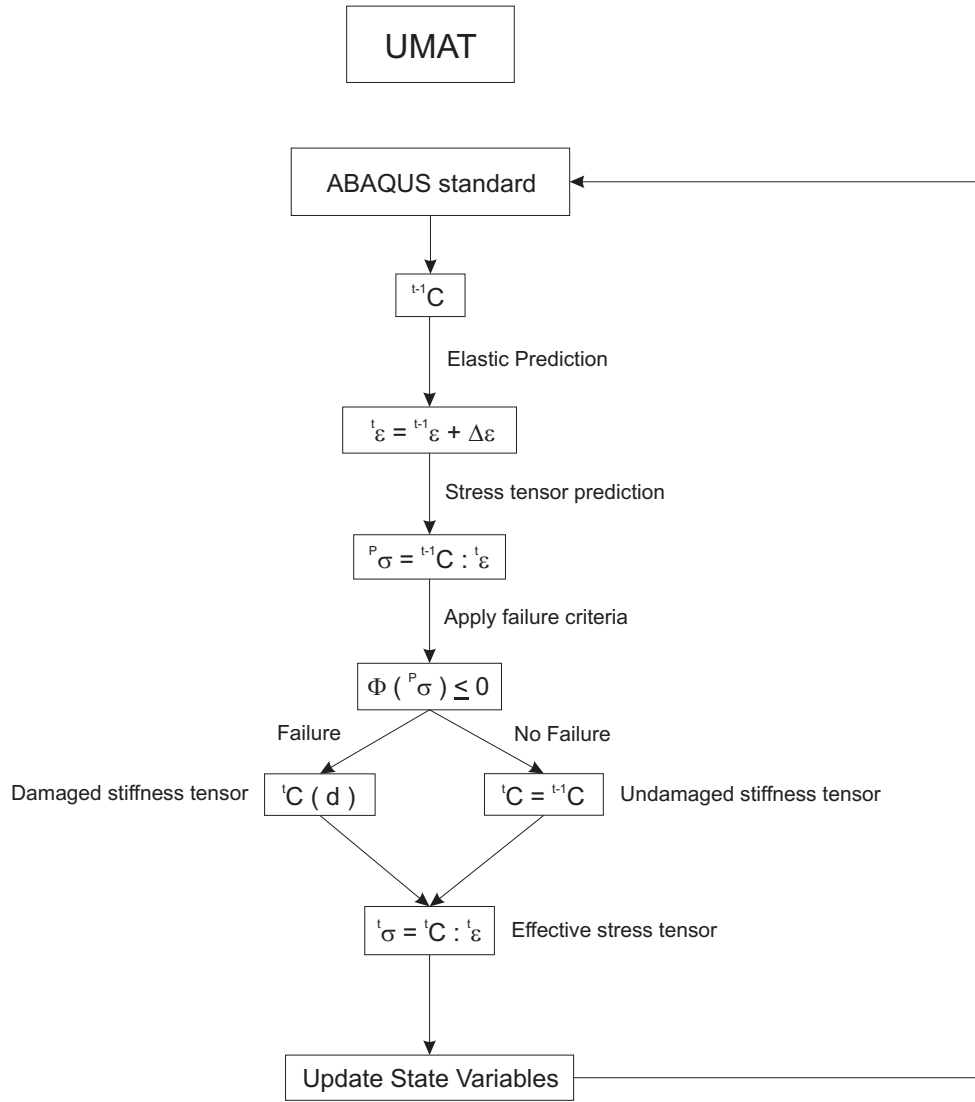


Figure 4-1: Algorithm of the UMAT user subroutine for ABAQUS standard.

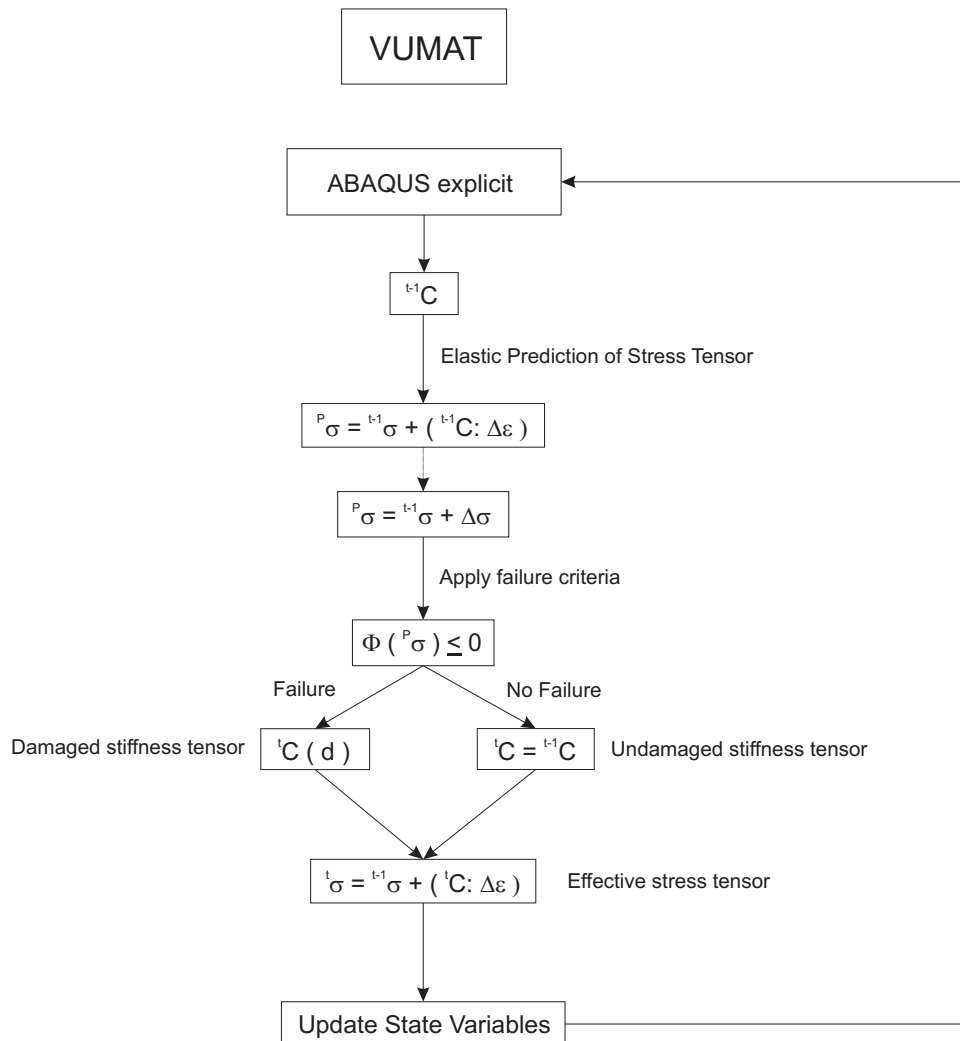


Figure 4-2: Algorithm of the VUMAT user subroutine for ABAQUS explicit.

The flowcharts of the proposed routines are shown in Figures 4-3 and 4-4.

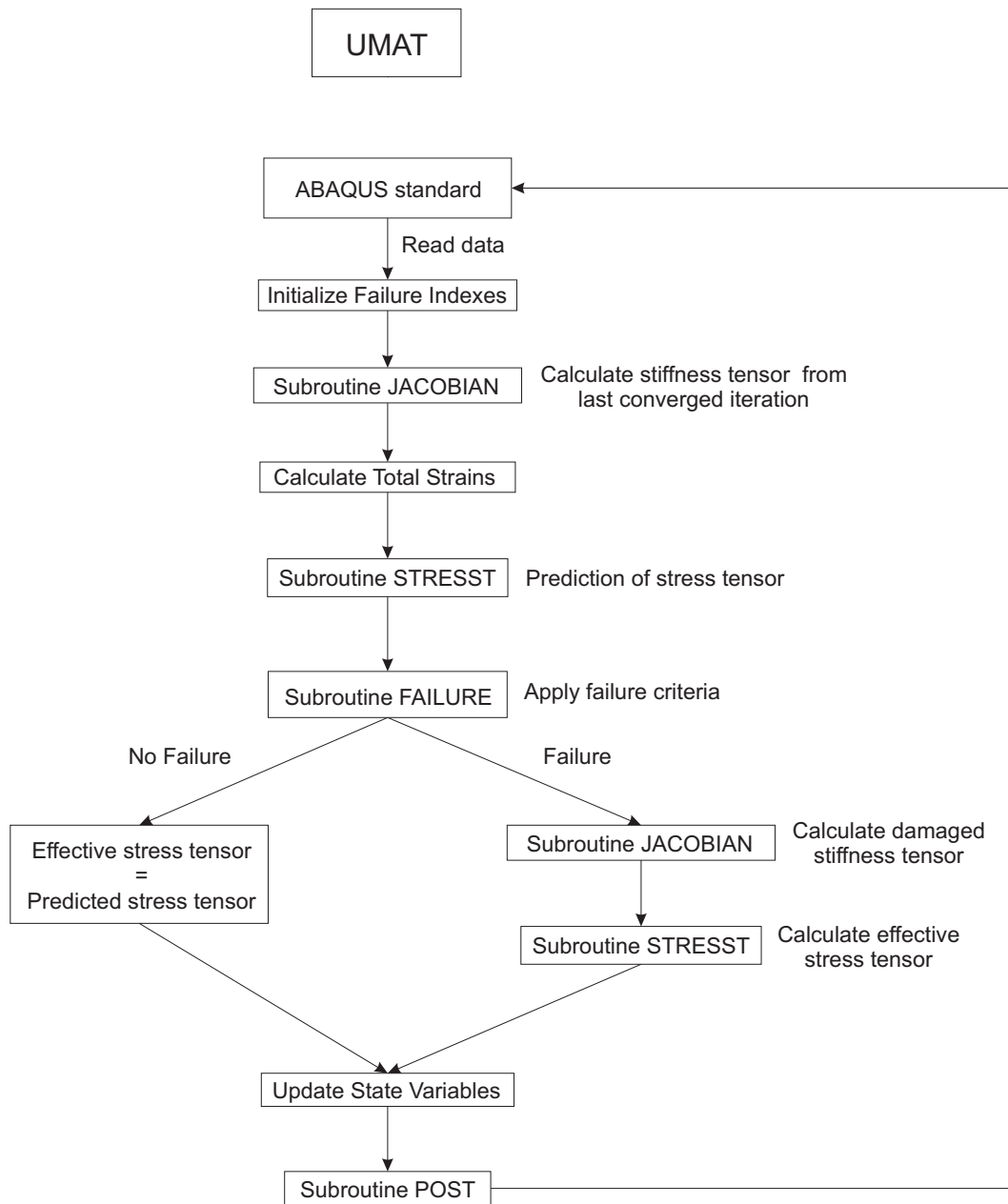


Figure 4-3: Flowchart of the UMAT user subroutine for ABAQUS standard.

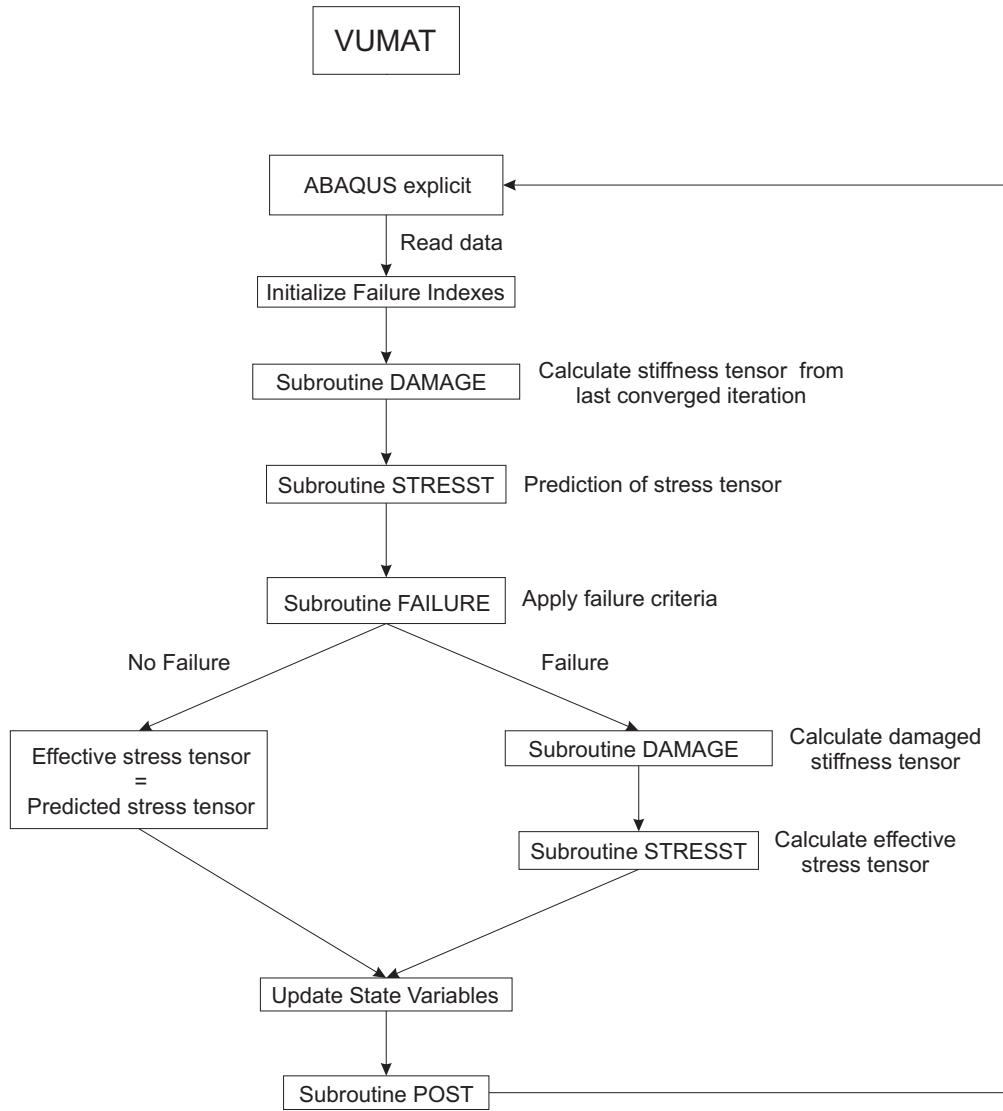


Figure 4-4: Flowchart of the VUMAT user subroutine for ABAQUS explicit.

The following sections of this chapter explain the proposed algorithms in detail.

4.1 Implicit model

The implicit model was implemented through an UMAT subroutine for usage with ABAQUS standard. This type of user subroutine implements a constitutive behaviour for a material that is not present in the standard material library of the program. The material model is simulated through the exchange of variables between the finite element code and the user subroutine. These variables will be described along the program flowchart shown in Figure 4-3.

A UMAT subroutine is called by ABAQUS standard one time per load increment. The UMAT is provided with the stress and strain tensors from the previous converged iteration, the deformation increment tensor for the current iteration and with the current values of the state variables indicating failure for each element in the last converged iteration. The UMAT must then compute the tangent stiffness tensor and the stress tensor for the current load increment, for each element using the corresponding material model. The procedure explained in this section is repeated for all elements each time the UMAT is called by the main program.

The first procedure is the initialization of the failure indexes to be used in the program. These indexes provide information to whether or not damage has occurred in the previous iteration. In case of previous damage occurrence these indexes state what material properties have previously been degraded and should remain degraded in the computation of the stiffness tensor for the next iteration.

After the initialization of the failure indexes, subroutine JACOBIAN is called for the first time to check for previous damage occurrence and compute the stiffness tensor from the last converged iteration (${}^{t-1}C$).

Using the strain tensor from the last converged iteration and the strain increment tensor, the total strain tensor for the current iteration is computed (${}^t\varepsilon = {}^{t-1}\varepsilon + \Delta\varepsilon$).

Subroutine STRESST is then called to compute a linear-elastic prediction of the stress tensor for the current iteration. This subroutine uses the stiffness

tensor from the last converged iteration and the previously computed total strain tensor (${}^P\sigma = {}^{t-1}C : {}^t\varepsilon$).

The failure criteria is then applied to the linear-elastic stress tensor prediction by calling subroutine FAILURE. The subroutine applies the LaRC03 failure criteria and updates an array of internal failure flags (FINDEX) indicating if any type of damage occurrence is predicted for the current iteration ($\Phi({}^P\sigma) \leq 0$).

After the application of the failure criteria, the program uses the FINDEX failure flags array to check for the occurrence of damage in the current iteration. If damage has occurred or has extended to more damage variables, the subroutine JACOBIAN is called a second time to compute the new stiffness tensor for the current iteration (${}^tC$), followed by a second call of subroutine STRESST to compute the effective stress tensor for the current iteration (${}^t\sigma = {}^tC : {}^t\varepsilon$).

If no damage occurrence or evolution was predicted by subroutine FAILURE in the current iteration, the linear-elastic stress tensor prediction is taken as the effective stress tensor (${}^t\sigma = {}^P\sigma$).

With the effective stress tensor and failure flags already defined, the program updates the state variables corresponding to the current iteration. The state variables assume the values of the failure flags. The process is then repeated for all elements.

Finally the effective stress tensor and the state variables for the current iteration are passed in for ABAQUS standard to perform the next load increment.

Two approaches were made with the implicit model. The dynamic and the static approaches which are detailed in the following subsections of this chapter.

The purpose of the two approaches done with the implicit model was to assess the influence of parameters such as model mass, loading speed and kinetic energy dissipation in the failure load and damage evolution predictions for the simulated specimens. Given the low density of the material and the low loading speed of the performed mechanical tests, a static approach where such parameters are not taken into account was also considered to be representative of the performed mechanical tests.

The comparison of the results obtained with the two approaches allows the user to identify the differences in representability and computational cost and then decide which approach to use with the implicit model.

4.1.1 Dynamic

The dynamic approach allows the user to accurately simulate the event of a mechanical test. A dynamic analysis step in ABAQUS standard accounts for the model mass and loading speed. When considering these factors, effects of energy dissipation peaks when damage occurrence is predicted and the material elastic properties are locally degraded become more evident, and have an effect on the load transfer to the non-degraded material zones.

A dynamic event simulation allows the user to assess the relevance of such effects when using the proposed continuum-damage model.

4.1.2 Static

Running a static analysis step in ABAQUS standard the user can simulate the event of a quasi-static mechanical test without considering the model mass or loading speed.

In the model definition for a static analysis the user specifies an applied load or displacement which will be applied to the model, and the progression rate of the loading application is controlled by the numerical convergence criteria of the main program.

As no kinetic energy or damping parameters are involved, sudden reductions of the model stiffness that occur when damage is predicted and the elastic material properties are degraded, originate convergence problems for the analysis. The convergence problems are related to the sudden reduction of the elastic properties, and to the associated discontinuity in the stress-strain relation. This leads to a severe time increment reduction or cutback which is required to achieve numerical convergence.

In a static analysis step with ABAQUS standard the user is allowed to specify an artificial damping parameter to reduce the effect of the discontinuities in the stress-strain relation. Using this parameter, numerical convergence rate is drastically improved and the usage of the proposed continuum-damage model becomes feasible.

4.2 Explicit model

The explicit model was implemented through a VUMAT subroutine for usage with ABAQUS explicit. Similarly to the UMAT, this type of user subroutine also implements a constitutive behaviour for a material that is not present in the standard material library of the program. The material model is also implemented through variable exchange with ABAQUS explicit, but the exchanged variables are different from those used with the UMAT. These variables will be described along the program flowchart shown in Figure 4-4.

A VUMAT subroutine is called by ABAQUS explicit one time per load increment. The VUMAT is provided with the stress tensor from the previous computed iteration, the strain increment tensor for the current iteration and with the current values of the state variables indicating failure for each element in the last computed iteration. The VUMAT must then compute the stress tensor and the internal variables to be used in the next load increment for each element using the corresponding material property. The procedure explained in this section is repeated for all integration points each time the VUMAT is called by the main program.

The first procedure is the initialization of the failure indexes to be used in the program. These indexes provide information to whether or not damage has occurred in the previous iteration. In case of previous damage occurrence these indexes state what material properties have previously been degraded and should remain degraded in the computation of the stiffness tensor for the next iteration.

Subroutine DAMAGE is then called for the first time to check for damage occurrence in the previous increment and to compute the stiffness tensor from the last increment according to the material previous damage state (${}^{t-1}C$).

After the computation of the stiffness tensor from the last increment, subroutine STRESST is called for the first time to compute a linear-elastic stress tensor prediction for the current increment, using the stiffness and the stress tensors from the last iteration and the strain increment tensor for the current iteration

$$({}^P\sigma = {}^{t-1}\sigma + ({}^{t-1}C : \Delta\varepsilon)).$$

The failure criteria is then applied to the linear-elastic predicted stress tensor by calling subroutine FAILURE. The subroutine applies the LaRC03 failure criteria and updates an array of internal failure flags (FINDEX) indicating if any type of damage occurrence is predicted for the current iteration ($\Phi({}^P\sigma) \leq 0$).

After the application of the failure criteria, the program uses the FINDEX failure flags array to check for the occurrence of damage in the current iteration. If damage has occurred or has extended to more failure variables, subroutine DAMAGE is called a second time to compute the new stiffness tensor for the current iteration (${}^tC$), followed by a second call of subroutine STRESST to compute the effective stress tensor for the current iteration (${}^t\sigma = {}^{t-1}\sigma + ({}^tC : \Delta\varepsilon)$).

If no damage occurrence or evolution was predicted by subroutine FAILURE in the current iteration, the linear-elastic stress tensor prediction is taken as the effective stress tensor (${}^t\sigma = {}^P\sigma$).

With the effective stress tensor and failure flags already defined, the program updates the state variables corresponding to the current iteration. The state variables assume the values of the failure flags.

The process is then repeated for all integration points.

Finally the effective stress tensor and the state variables for the current iteration are passed in for ABAQUS explicit to perform the next load increment.

With the explicit model the only possible approach is a dynamic step analysis. Due to the explicit time integration scheme, a maximum stable time step is defined by ABAQUS explicit based on the stiffness and mass properties of the simulation model.

The material studied with the proposed continuum-damage model has a very low density and a very high specific stiffness. The explicit time integration scheme used by the analysis software is based on the central differences method. The discrete mass matrix of the model has a very important role in the integration process and on the definition of the maximum stable time increment, and the described characteristics of the chosen material cause the maximum stable time

increment to be very small. If nothing is done to change this, huge computational cost for a relatively simple analysis would occur.

To avoid these problems ABAQUS explicit allows the user to perform a "*mass scaling*" operation. This operation consists in artificially increasing the density of the material to balance the relation between density and stiffness of the material, therefore increasing the value of the maximum stable time increment.

When using the proposed "*mass scaling*" option the user must be careful not to disturb the dynamic nature of the simulation.

Abaqus [1] suggests that a "*mass scaling factor*" is considered appropriate as long as the value of the artificial energy of the model never rises above 0.5% of the internal energy at any given time.

Using the stated recommendation the usage of the explicit integration technique improves the computational cost of the model.

Chapter 5

Experimental Tests

5.1 Introduction

The usage of a continuum-damage model applied to FE simulations requires experimental validation. The experimental tests performed for results comparison purposes are described in this section.

5.2 Test matrix

The material used for the test specimens was the Hexcel IM7/8552 carbon epoxy unidirectional pre-preg characterized in Chapter 2 of this work.

The test specimens are composite laminates with a $(90^\circ/0^\circ/45^\circ/-45^\circ)_s$ stacking sequence.

The tests were performed according to the test matrix shown in Table 5.1, following ASTM standard D-5766 [22].

Two parameters the hole diameter, d and the ratio between the specimen width and the hole diameter, w/d , were considered. Three levels for each parameter were used ($d=6, 8, 10\text{mm}$ and $w/d=2, 4, 6$). The consideration of the different levels for the w/d parameter aims to assess the accuracy of the continuum-damage model, in predicting deterministic size effects which result from testing specimens with different hole sizes. Five specimens were tested for each configuration, using

Table 5.1: Open-hole tensile test matrix.

Spec. Ref.	d	w	w/d	Condition	# of specimens
OHT1	6	12	2	RT/D	5
OHT2	6	24	4	RT/D	5
OHT3	6	36	6	RT/D	5
OHT4	8	16	2	RT/D	5
OHT5	8	32	4	RT/D	5
OHT6	8	48	6	RT/D	5
OHT7	10	20	2	RT/D	5
OHT8	10	40	4	RT/D	5
OHT9	10	60	6	RT/D	5

an MTS servo-hydraulic test machine.

The central hole of the specimens was examined using X-ray images to assess the machining quality. Figures 5-1 and 5-2 show the X-ray images for some of the tested specimens.

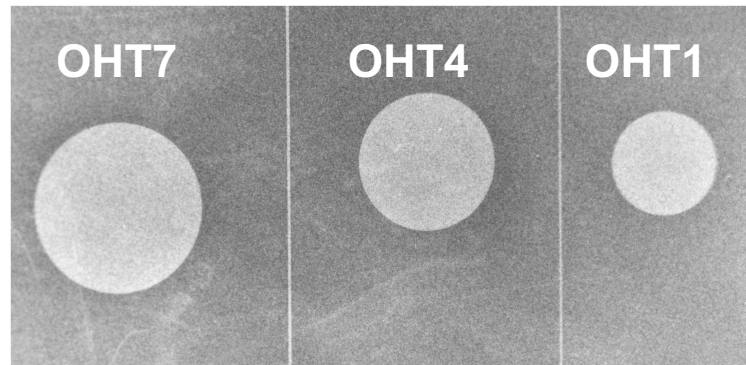


Figure 5-1: X-ray results.

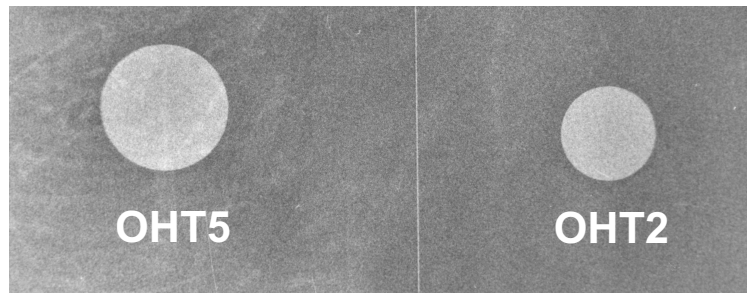


Figure 5-2: X-ray results.

The above presented Figures show that no delamination was present in the hole vicinity for all the specimens.

5.3 Test results

Figures 5-3 to 5-11 show the load-cross head displacement relation obtained for all the geometries considered.

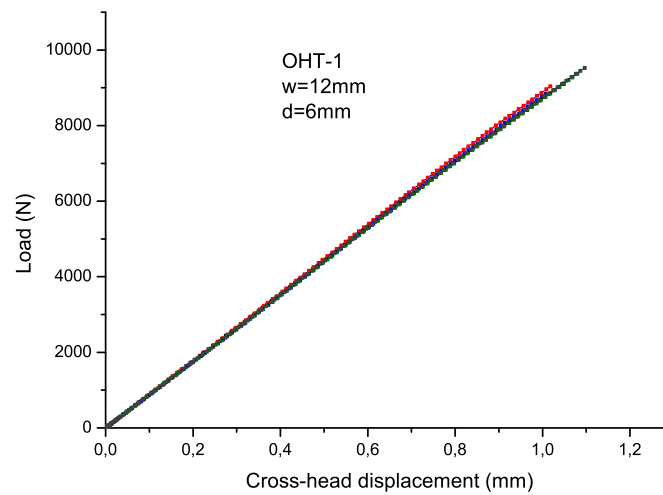


Figure 5-3: Load-cross head displacement for the specimens OHT1.

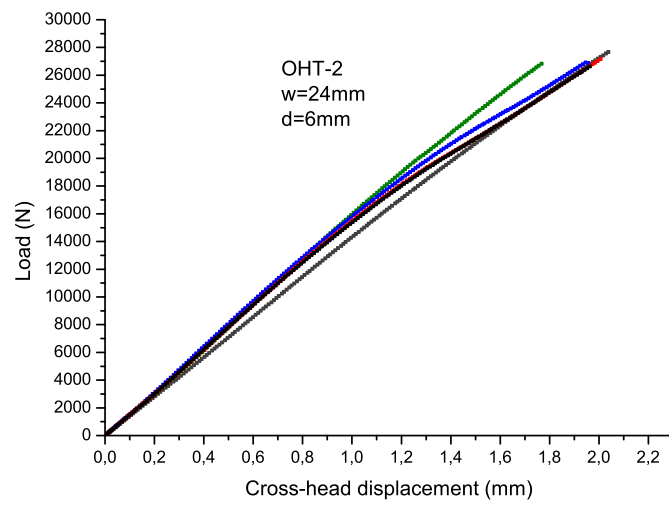


Figure 5-4: Load-cross head displacement for the specimens OHT2.

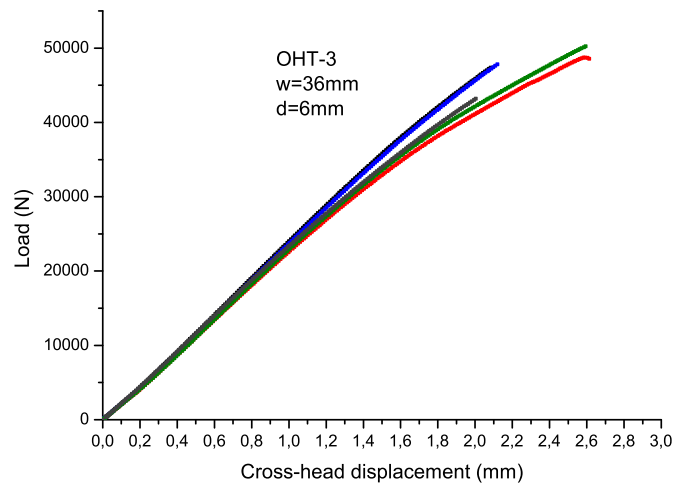


Figure 5-5: Load-cross head displacement for the specimens OHT3.

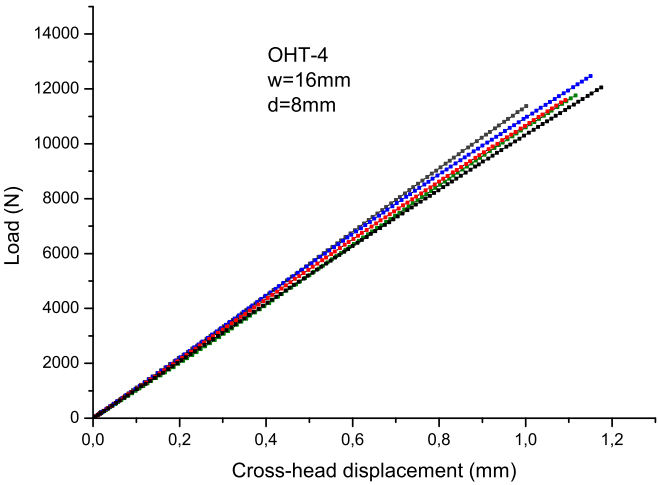


Figure 5-6: Load-cross head displacement for the specimens OHT4.

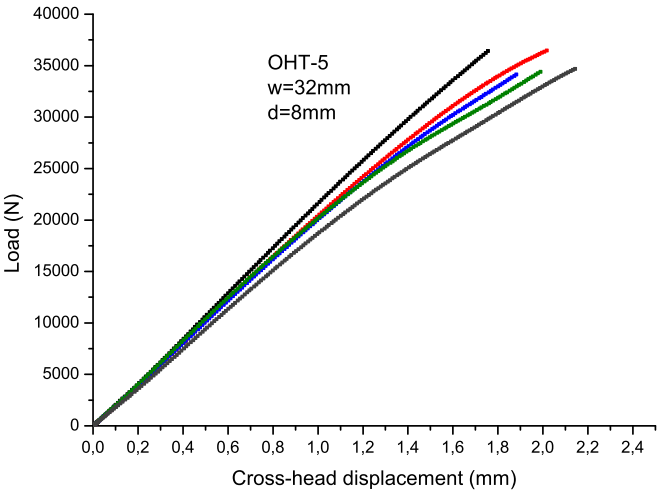


Figure 5-7: Load-cross head displacement for the specimens OHT5.

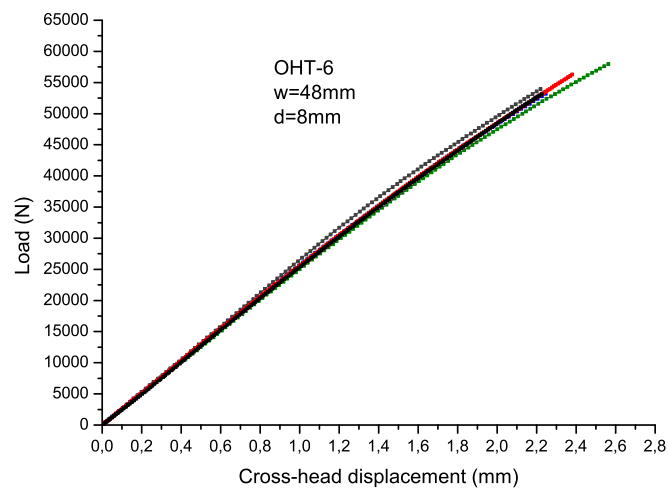


Figure 5-8: Load-cross head displacement for the specimens OHT6.

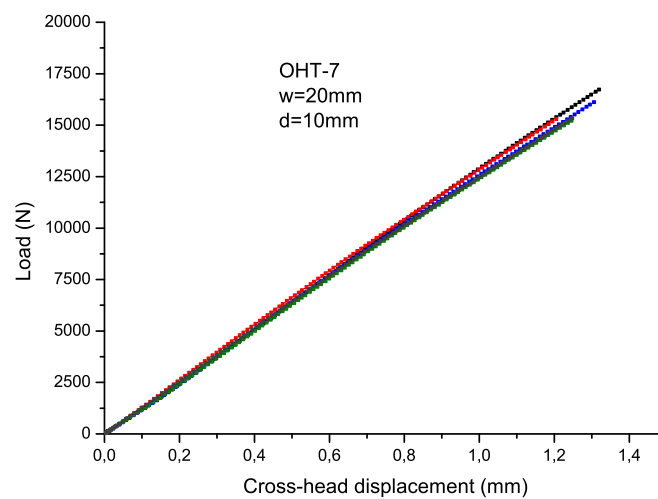


Figure 5-9: Load-cross head displacement for the specimens OHT7.

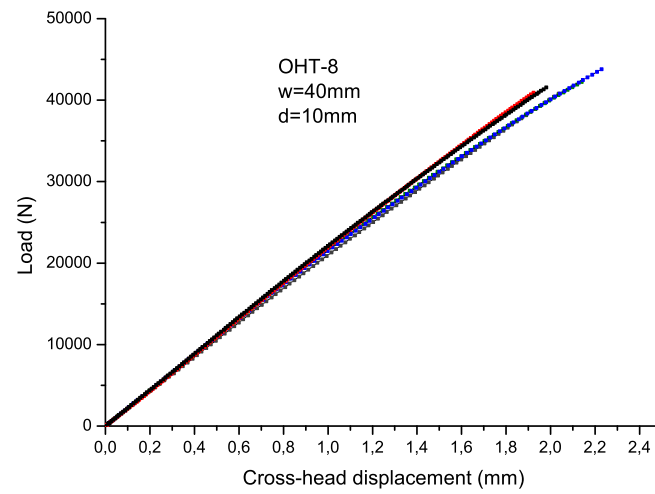


Figure 5-10: Load-cross head displacement for the specimens OHT8.

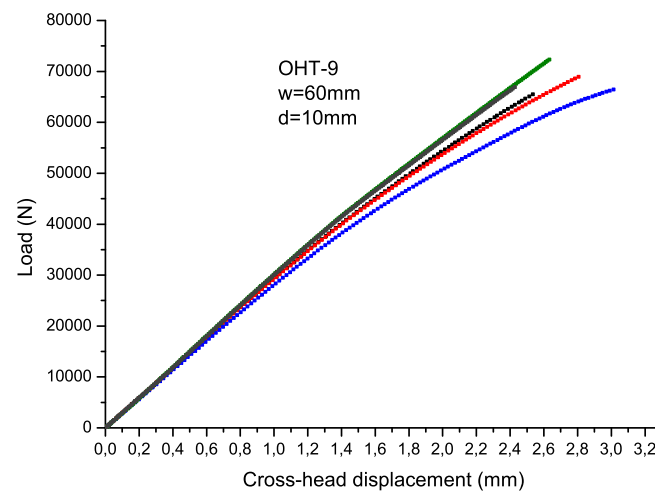


Figure 5-11: Load-cross head displacement for the specimens OHT9.

Tables 5.2 to 5.10 present the maximum load ($\bar{P}$) and the remote stress at failure ($\bar{\sigma}^\infty = \bar{P}/A$) for all the specimens tested.

Table 5.2: Results of open-hole tensile test: specimen OHT1.

Spec. Ref.	$\bar{P}$ (N)	$\bar{\sigma}^\infty$ (MPa)
OHT1-1	9100	249.86
OHT1-2	9040	247.61
OHT1-3	8848	242.54
OHT1-4	9372	256.91
OHT1-5	9524	261.94
Average	9176.8	252.2
STDV	269.9	8.8
CV (%)	2.9	3.5
IC	± 335.1	± 10.9

Table 5.3: Results of open-hole tensile test: specimen OHT2.

Spec. Ref.	$\bar{P}$ (N)	$\bar{\sigma}^\infty$ (MPa)
OHT2-1	27676	385.35
OHT2-2	26832	368.98
OHT2-3	26916	370.12
OHT2-5	27152	374.61
OHT2-6	26632	364.72
Average	27041.6	372.76
STDV	400.7	7.9
CV (%)	1.5	2.1
IC	± 497.4	± 9.8

Table 5.4: Results of open-hole tensile test: specimen OHT3.

Spec. Ref.	P (N)	$\bar{\sigma}^{\infty}$ (MPa)
OHT3-1	47384	438.74
OHT3-2	48720	449.61
OHT3-3	47824	442.81
OHT3-4	50244	465.09
OHT3-5	43192	397.17
Average	47472.8	438.7
STDV	2631.1	25.3
CV (%)	5.5	5.8
IC	± 3266.5	± 31.4

Table 5.5: Results of open-hole tensile test: specimen OHT4.

Spec. Ref.	P (N)	$\bar{\sigma}^{\infty}$ (MPa)
OHT4-1	11372	237.32
OHT4-2	11764	244.92
OHT4-3	12472	259.70
OHT4-4	11600	241.42
OHT4-5	12152	253.35
Average	11872.0	247.3
STDV	439.9	9.1
CV (%)	3.7	3.7
IC	± 546.1	± 11.3

Table 5.6: Results of open-hole tensile test: specimen OHT5.

Spec. Ref.	P (N)	$\bar{\sigma}^{\infty}$ (MPa)
OHT5-1	36428	378.20
OHT5-2	36468	378.97
OHT5-3	34144	356.11
OHT5-4	34392	358.36
OHT5-5	34692	361.49
Average	35224.8	366.6
STDV	1133.4	11.1
CV (%)	3.2	3.0
IC	± 1407.1	± 13.8

Table 5.7: Results of open-hole tensile test: specimen OHT6.

Spec. Ref.	P (N)	$\bar{\sigma}^{\infty}$ (MPa)
OHT6-1	53948	368.11
OHT6-2	57976	395.60
OHT6-3	53196	364.40
OHT6-4	56284	387.95
OHT6-5	53120	362.46
Average	54904.8	375.7
STDV	2142.2	15.1
CV (%)	3.9	4.0
IC	± 2659.4	± 18.7

Table 5.8: Results of open-hole tensile test: specimen OHT7.

Spec. Ref.	P (N)	$\bar{\sigma}^{\infty}$ (MPa)
OHT7-1	16728	276.95
OHT7-2	15292	253.18
OHT7-3	16124	266.95
OHT7-4	15220	252.82
OHT7-5	15332	253.84
Average	15739.2	260.8
STDV	663.5	10.8
CV (%)	4.2	4.2
IC	± 823.8	± 13.4

Table 5.9: Results of open-hole tensile test: specimen OHT8.

Spec. Ref.	P (N)	$\bar{\sigma}^{\infty}$ (MPa)
OHT8-1	40812	335.12
OHT8-2	42212	347.14
OHT8-3	43752	359.80
OHT8-4	40920	335.41
OHT8-5	41552	345.12
Average	41849.6	344.5
STDV	1202.1	10.2
CV (%)	2.9	2.9
IC	± 1492.3	± 12.6

Table 5.10: Results of open-hole tensile test: specimen OHT9.

Spec. Ref.	P (N)	$\bar{\sigma}^\infty$ (MPa)
OHT9-1	65496	359.08
OHT9-3	68948	378.00
OHT9-4	66452	364.20
OHT9-5	72336	395.28
OHT9-6	66920	371.72
Average	68030.4	373.7
STDV	2717.0	14.1
CV (%)	4.5	3.8
IC	± 3373.0	± 17.5

The failure mode observed for all specimens was net-section tension, as shown in Figure 5-12.



Figure 5-12: Net-section failure in an open-hole specimen.

5.4 Acoustic emission results

The use of acoustic emission techniques can provide important insight concerning the damage mechanisms occurring in the open-hole test specimens. The data produced with this technique can be compared with the damage evolution predictions given by the continuum-damage model and be used to validate its behaviour. One specimen of each geometry was instrumented with strain gages and with two piezoelectric acoustic emission sensors. The strain-gauge measurements, not presented in this section to avoid redundancy, will be used in Chapter 7 for a direct comparison with the FE simulations results using the continuum-damage model. Figure 5-13 shows one specimen instrumented.

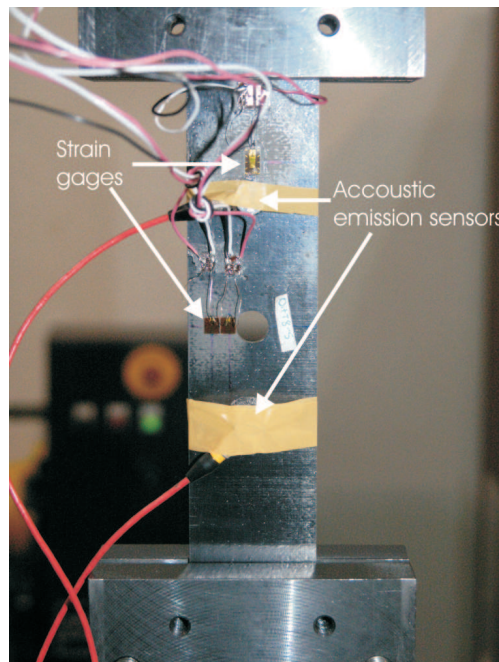


Figure 5-13: Specimen OHT8 instrumented.

Acoustic emission (AE) is defined as the class of phenomena whereby transient elastic waves are generated by the rapid release of energy from localized sources within a material, or the transient elastic wave(s) so generated. AE is a nondestructive technique that allows the detection, in real-time, of evolutive defects. AE is the phenomenon of emission and propagation of stress waves resulting from local modifications (e.g. cracking, dislocations) in a material submitted to

a mechanical loading. AE results from local rupture processes, produced when a part of the strain energy stored in the material is rapidly released, creating a temporal discontinuity of the deformation called AE event. The AE event generates a mechanical wave in the structure. In the case of composites materials, AE is promising because those materials emit high amplitude signals due to their brittleness, anisotropy and heterogeneity [23]. The damage mechanisms sources of AE are matrix cracking, fibre-matrix interface debonding, fibre fracture, and delamination. AE results enable the knowledge of the state of the extended volume. This technique is one of the rare allowing the detection and initiation of defects and the observation of propagation of those defects, in real-time.

For AE waves acquisition a two-channels AMSY-5 AE system developed by Vallen-System GmbH was used. The signals were digitalized by an analogue/digital converter having a dynamic range of 16 bit using a sampling rate of 10 MHz. A resonant piezoelectric transducer, Micro 30 of Physical Acoustics Corporation and a broadband B1025 piezoelectric transducer of DigitalWave Corporation, with a near flat frequency response from 150 kHz to 2 MHz, were used. The transducers outputs are amplified using two preamplifiers permitting the AE signals to have more usable voltage. The signals are then transmitted over a long length of BNC cable allowing the AE system to be distant from the testing machine. The preamplifiers provide a gain of 100 (40 dB) and have plug-in frequency filters to eliminate the mechanical noise and background noise that prevails at low frequency.

In this work two preamplifiers 1220A from Physical Acoustics Corporation with a 40 dB gain and a high pass 100 kHz plug-in filter were used. The location of AE wave's sources was derived from the arrival time difference of AE waves to the transducers. The transducers were mounted on the specimen using a tape. Silicone grease was used as coupling agent between the sensor and the composite surface. A pre-processing of the AE signals was performed in order to remove from the recorded signals the unavoidable noise (e.g. electromagnetic interference, hydraulic, vibration, fretting noise). The AE activity was measured during the

tests as a function of the applied load. Damage initiation can be determined by the increase of AE activity observed in the Cumulative AE events vs Time curves.

Figures 5-14 to 5-22 show the relation between the load and the AE events for all the geometries considered.

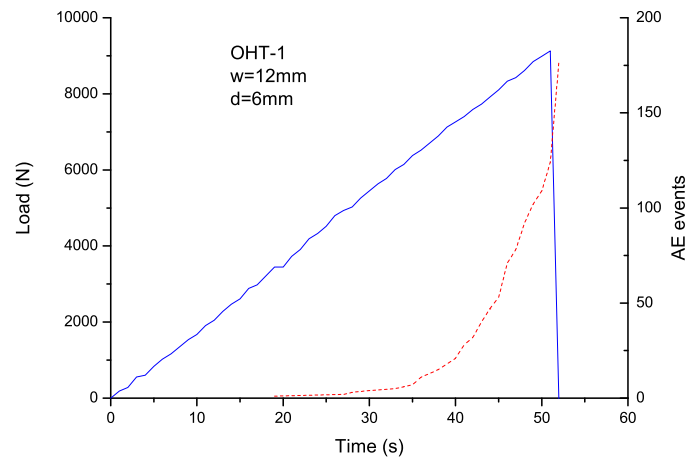


Figure 5-14: Load-AE relation for the specimen OHT1.

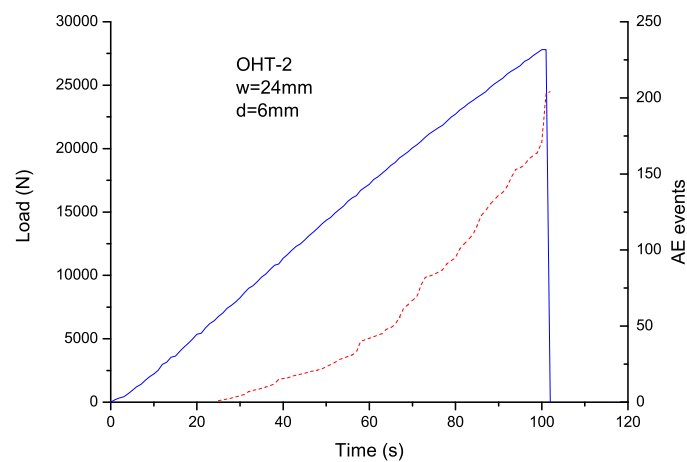


Figure 5-15: Load-AE relation for the specimen OHT2.

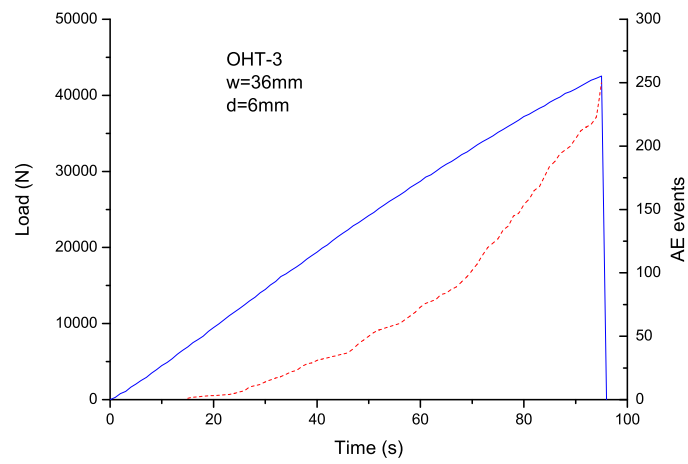


Figure 5-16: Load-AE relation for the specimen OHT3.

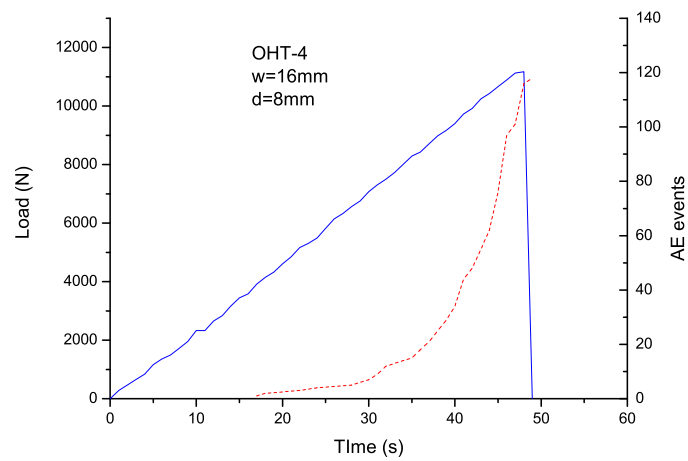


Figure 5-17: Load-AE relation for the specimen OHT4.

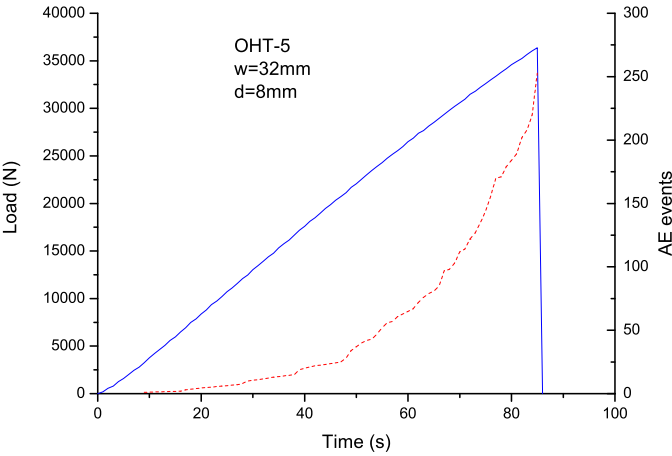


Figure 5-18: Load-AE relation for the specimen OHT5.

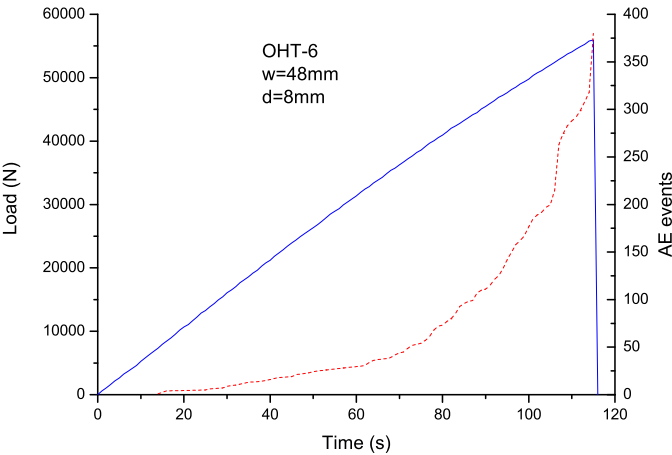


Figure 5-19: Load-AE relation for the specimen OHT6.

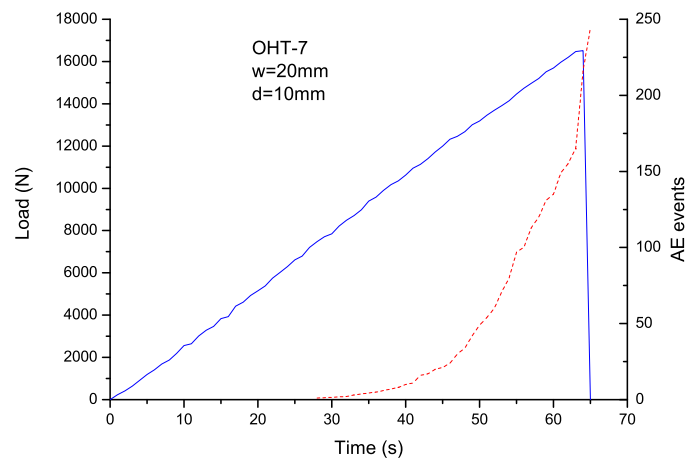


Figure 5-20: Load-AE relation for the specimen OHT7.

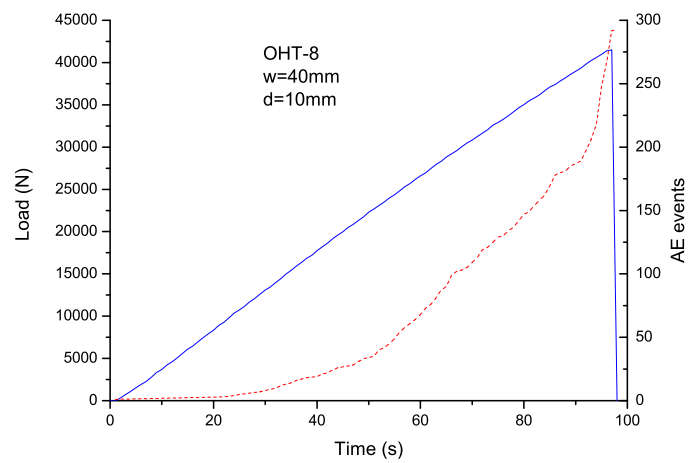


Figure 5-21: Load-AE relation for the specimen OHT8.

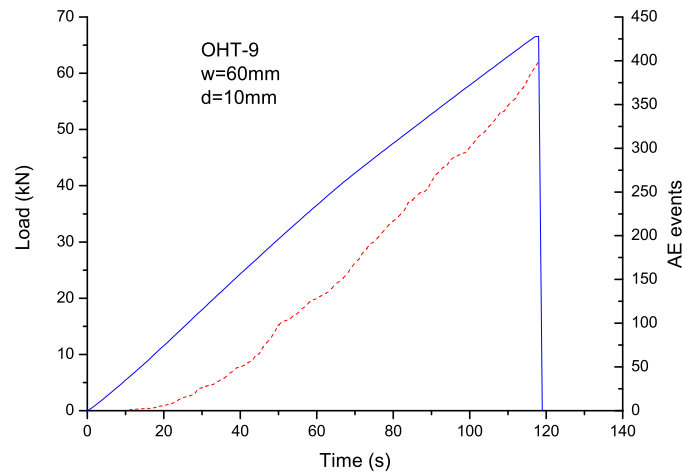


Figure 5-22: Load-AE relation for the specimen OHT9.

From the analysis of the previous Figures it can be concluded that damage, i.e. matrix cracking, occurs at loads much lower than the ultimate failure load of the test specimen. The non-critical damage mechanisms do not significantly affect the load-time (or cross-head displacement) relation.

Damage initiation was determined from the AE activity. Damage onset was detected by an important increase of the rate (i.e. number of events detected per unit of time) of AE. This can be observed in the AE events vs. Time curves. Considering Figure 5-23, it can be seen that the typical AE events vs. Time curves for the OHT specimens were constituted of three phases.

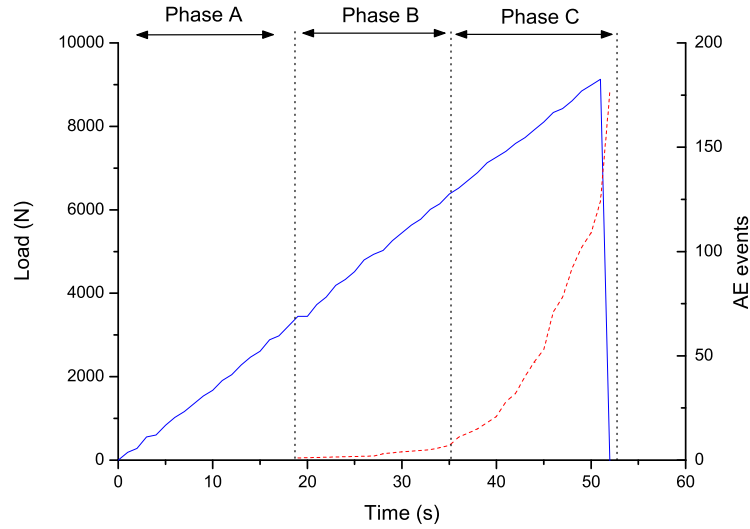


Figure 5-23: Different phases of AE in the open-hole test specimens.

- Phase A, corresponding to a silent phase where no damage occurs.
- Phase B, characterized by the appearance of the first AE events. The detected signals have relatively low amplitude (i.e. maximum amplitude) and low duration. The sources of the AE events are randomly located between the two sensors. The signals are thus thought to be due to matrix cracking.
- Phase C, characterized by a change of the slope of the cumulative AE events curve. This represents the occurrence of other damage mechanisms such as fibre/matrix decohesion, delamination and fibre fracture successively. Fibres fractures are detected at the final part of the test. This is represented by high amplitude signals recorded just at the specimens' fracture.

The loads corresponding to the onset of damage in all the specimens tests are shown in Table 5.11.

Table 5.11: Damage onset loads for the open-hole test specimens (values in kN).

Specimen Ref.	Damage onset load, P^o	Failure load, $\bar{P}$	$\frac{P^o}{\bar{P}} \times 100$
OHT1	6.2	9.2	67.4
OHT2	13.8	27.0	51.1
OHT3	17.2	47.5	36.2
OHT4	8.5	11.9	71.4
OHT5	20.8	35.2	59.1
OHT6	35.9	54.9	65.4
OHT7	11.7	15.7	74.5
OHT8	21	41.8	50.2
OHT9	20	68.0	29.4

Chapter 6

Finite Element Models

6.1 Introduction

All the models of the specimens used for validation use the same general structure varying only in terms of geometry to meet the test specimens specifications, and specific material properties to meet the implicit static, implicit dynamic and explicit cases. The modeling description presented in this section is therefore valid for all specimens.

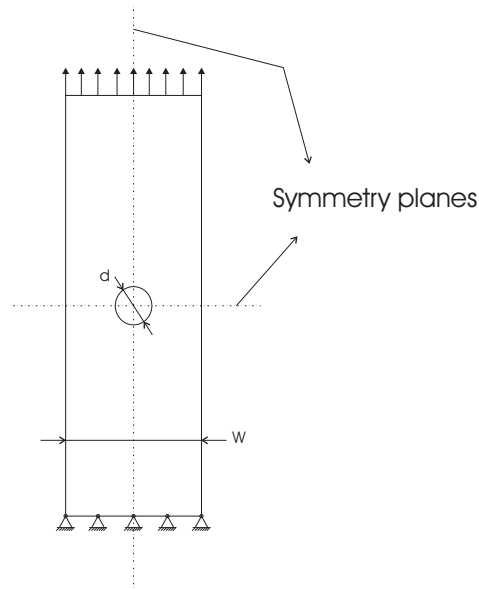


Figure 6-1: Case study illustration.

The general case study is presented in Figure 6-1 where geometry symmetry planes are also identified. The dimensions W and d represent respectively the width of the plate and the central hole diameter that follow the parameters of the experimental test specimens.

6.2 Boundary conditions

In Figure 6-1 is shown that the specimens to be modeled present two geometric symmetry planes. The finite element models were defined for a quarter of the geometry (Figure 6-2) taking advantage of those planes to reduce computational cost.

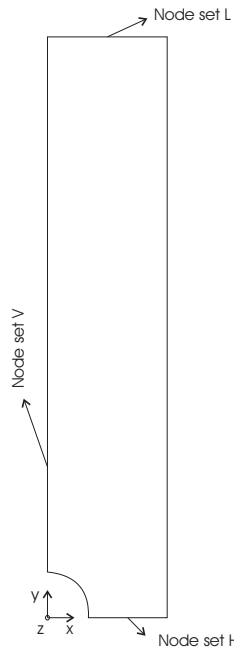


Figure 6-2: Modeled geometry illustration.

The boundary conditions applied to the finite element models were symmetry boundary conditions. The constrained DOFs (degrees of freedom) for node sets H and V shown in Figure 6-2 are schematically presented in Table 6.1.

For a better understanding of the applied boundary conditions, Figure 6-3 shows the hole vicinity area of a constrained specimen.

The meshing strategy for all the models was to refine the hole vicinity area

Table 6.1: DOF constraint scheme for the selected node sets.

Node set	U _x	U _y	U _z	UR _x	UR _y	UR _z
H	free	constrained	free	constrained	free	constrained
V	constrained	free	free	free	constrained	constrained

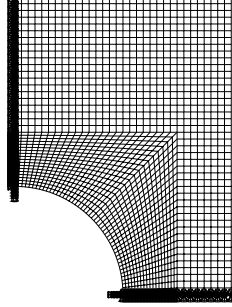


Figure 6-3: Boundary conditions detail in the surrounding hole area.

in order to improve the damage initiation predictions. Figure 6-3 also illustrates a typical mesh, with partitions around the hole in order to avoid excessively distorted elements.

6.3 Loading conditions

The loading conditions were applied to simulate the mechanical test. A displacement was applied to the node set L shown in Figure 6-2 in the positive y axis direction.

For the dynamic analysis step models the specimen loading speed of 2mm/min used in the mechanical testing was simulated through the *AMPLITUDE keyword defined in the analysis input file. This keyword allows the user to specify the application rate of the displacement, defined in terms of magnitude and time to meet the specifications of the mechanical test event.

For the static analysis step models the loading speed is irrelevant since there is no mass or inertia present. A simple displacement load was applied. The application rate of the displacement in these cases is controlled by Abaqus numeric

convergence parameters and varies through the analysis. As mentioned before the models simulate a quasi-static mechanical testing event, so the fact that the loading speed is not contemplated should not be relevant.

6.4 Element properties

The finite element models presented in this work use only composite shell elements. These elements were chosen due to their adequacy to the case-study. The shell elements are computationally less expensive than three dimensional models and compute the mechanical behaviour for all the laminate plies using a single element through the laminate thickness.

For the implicit models standard S4 Abaqus elements were used. These are four node complete integration shell elements. For the explicit models and only due to ABAQUS explicit limitations, standard Abaqus S4R elements were used. These are four node reduced integration shell elements.

Both elements used allow the user to fully specify the number of plies, stacking sequence and ply thickness providing an adequate representation of the material in analysis.

Using the referred elements the user has access to contour plots independently indicating the value of each damage variable for each laminate ply. The information can be used to monitor damage initiation and evolution, as well as the damage mechanisms involved.

In all cases the proposed continuum-damage model is implemented as a material constitutive behaviour model. In order to reduce computational cost the elements of the specimen in the area corresponding to the clamped area in the experimental test specimens were modeled with standard Abaqus laminate composite constitutive behaviour.

Table 6.2 states the material properties ¹ necessary to run an analysis with the

¹All the material property symbols presented in Table 6.2 are described in the List of Symbols.

proposed continuum-damage model (CDM) and with standard Abaqus laminate composite constitutive behaviour ².

Table 6.2: Required material properties.

Material property	CDM	Abaqus
E_1	✓	✓
E_2	✓	✓
ν_{12}	✓	✓
G_{12}	✓	✓
G_{13}	-	✓
G_{23}	-	✓
X_T	✓	-
X_C	✓	-
Y_T^{is}	✓	-
Y_C	✓	-
S_L^{is}	✓	-
S_T	✓	-
η^L	✓	-
η^T	✓	-
φ	✓	-
g	✓	-
α_1	✓	✓
α_2	✓	✓

²Note that Table 6.2 presents only material properties. Parameters as property degradation coefficients and working temperature, also necessary to run an analysis, are not in the scope of this Table

Chapter 7

Examples and Comparison with Experimental Results

7.1 Introduction

In this chapter the results obtained with the proposed continuum-damage model in its various approaches are compared with experimental results. The analysis of the results is used to validate the proposed model predictions.

As described in Chapter 5 the specimens tested and modeled correspond to CFRP laminate plates containing a central hole. The geometry specifications for each specimen are described in Table 5.1.

The comparison of the simulations using the implicit models and the experimental data is presented in terms of load-deformation graphics using the strain gauges test data, damage evolution using the acoustic emission test data, and ultimate failure load predictions.

It was not possible to compute all of the simulations with the explicit model during the preparation of this work. The considerable computational cost of these simulations allowed the computation of only five of the initially proposed nine simulations. However, the results obtained with the simulations using the explicit model within the scope of another ongoing work of the author, was enough to present fundamental conclusions that are discussed in Chapter 8 of this work.

For the explicit models it is not possible to include a load-deformation curve superimposed to the experimental and implicit model curves. This is due to a limitation of the finite element code used, which only allows the user to obtain logarithmic deformation data for the desired elements. The explicit model results are presented in terms of damage evolution on a qualitative basis due to the above mentioned limitation, and ultimate failure load predictions.

7.2 Load-deformation

The load-deformation curves presented were obtained experimentally using strain gauges data, and numerically using the average deformation at the integration points corresponding to the area of influence of the strain gauges at the external ply of the simulated laminate.

Table 7.1 shows the position of the strain gauges used, relatively to the general coordinates-system illustrated in Figure 7-1.

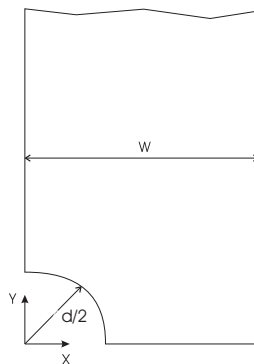


Figure 7-1: General coordinates-system for the strain gauges position.

Table 7.1: Strain gauges position.

Spec. Ref.	$x(mm)$	$y(mm)$
OHT1	4.5	0
OHT2	5.5	0
OHT3	5.5	0
OHT4	6.0	0
OHT5	6.5	0
OHT6	8.5	0
OHT7	7.5	0
OHT8	7.5	0
OHT9	7.5	0

Figures 7-2 to 7-10 show the load-deformation curves for all the specimens using the implicit dynamic and the implicit static models.

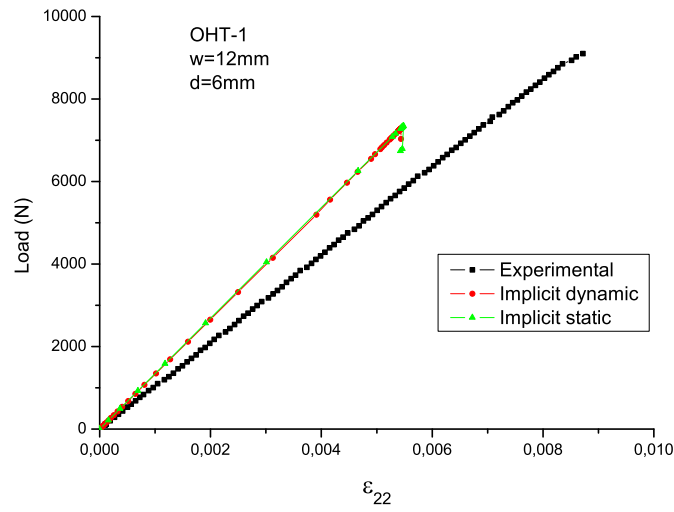


Figure 7-2: Load-deformation curve for the specimen OHT1.

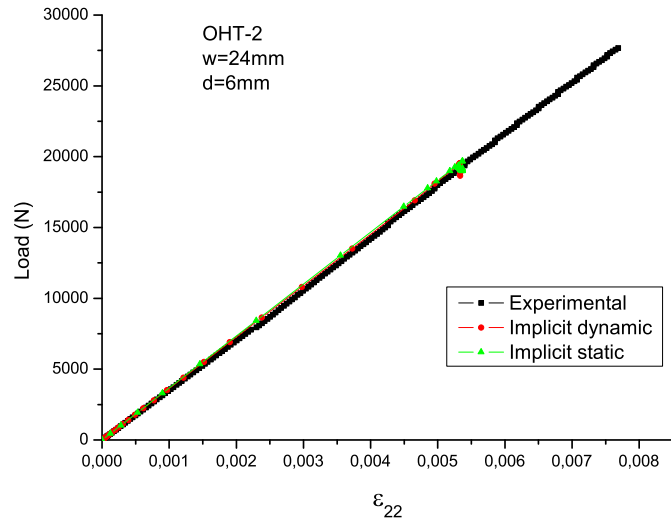


Figure 7-3: Load-deformation curve for the specimen OHT2.

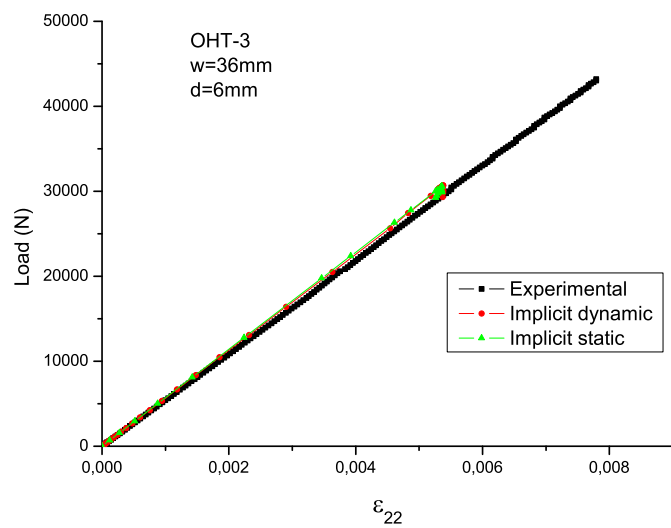


Figure 7-4: Load-deformation curve for the specimen OHT3.

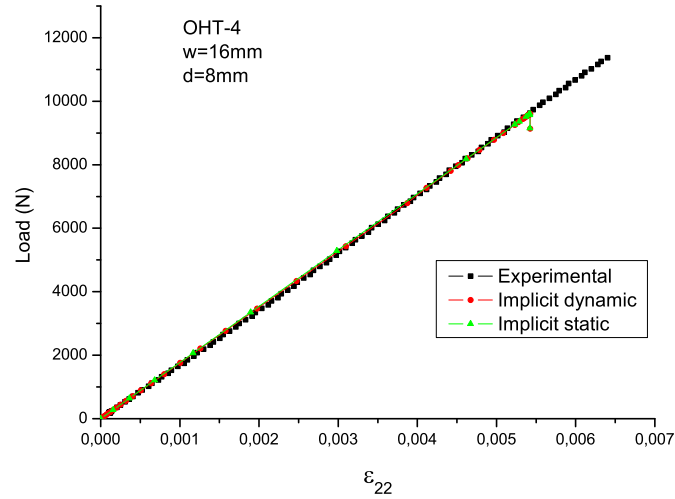


Figure 7-5: Load-deformation curve for the specimen OHT4.

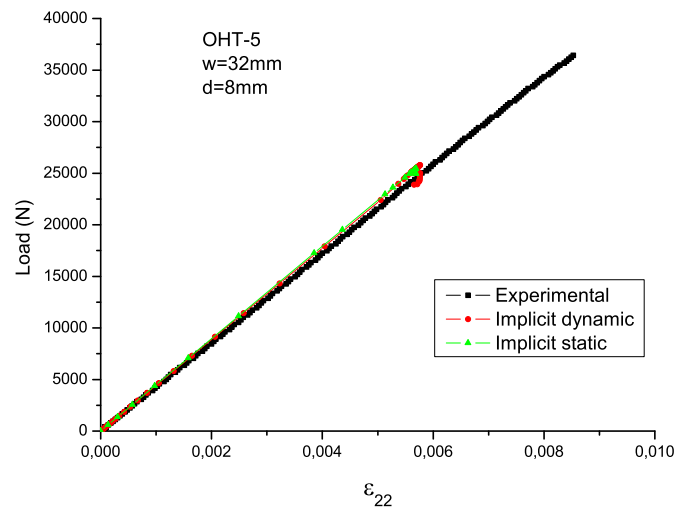


Figure 7-6: Load-deformation curve for the specimen OHT5.

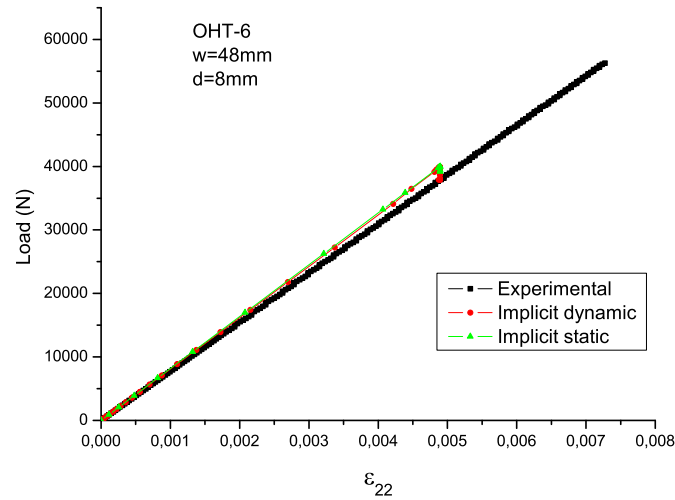


Figure 7-7: Load-deformation curve for the specimen OHT6.

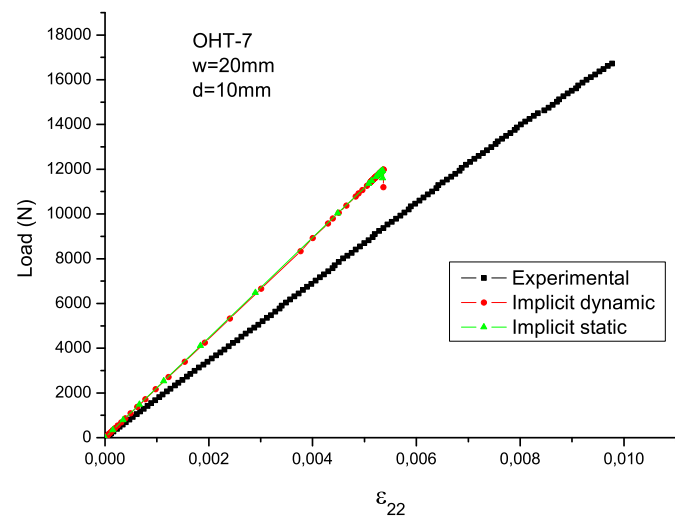


Figure 7-8: Load-deformation curve for the specimen OHT7.

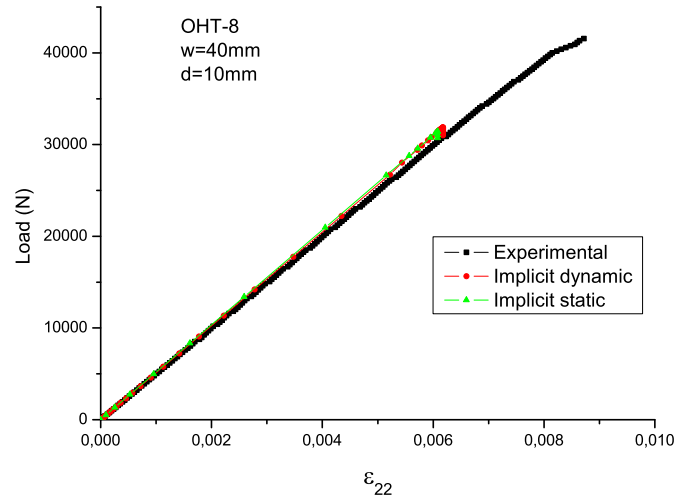


Figure 7-9: Load-deformation curve for the specimen OHT8.

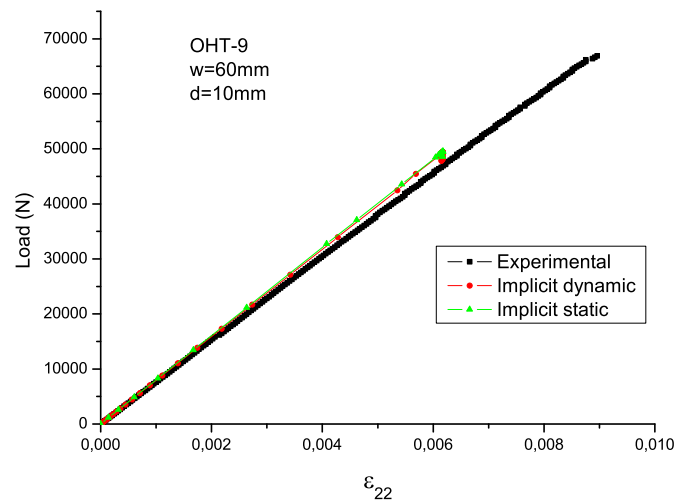


Figure 7-10: Load-deformation curve for the specimen OHT9.

The numerically computed specimen stiffness correlates well to the experimental stiffness for all specimens but OHT1 (Figure 7-2) and OHT7 (Figure 7-8). The offset in the load-deformation curves for the referred specimens is due to the low w/d ratio, meaning that the boundary proximity effect that originates a very high stress gradient in the strain gauges fixation point for these specimens, combined with the integration points placement approximation used had a greater influence on the results than for all the other specimens.

Note that the curves presented in Figures 7-2 to 7-10 are plotted up to the strain gauges failure point, which occurs at a lower load value than the ultimate failure load and corresponds to the failure of the laminate's external ply at the strain gauges fixation point. The final observation to make is that the numeric predictions of strain gauge failure (external ply failure at the strain gauge fixation point) are consistently lower than the experimental strain gauge failure load. This is related to the stress distribution associated with a sudden reduction of the elastic properties which increases the load-transfer effect to the undamaged element promoting a slightly premature failure prediction for the integration points used.

7.3 Damage evolution

As discussed in Section 5.3 of this work, the acoustic emission test data reveals three distinct phases for the damage evolution in the test specimens. Figure 5-23 shows the typical damage evolution in the tested specimens.

From the acoustic emission test data it can be concluded that the damage evolution follows the same sequence for every specimen tested. To avoid redundancy only the OHT3 specimen will be used for comparison purposes in this Section.

Figures 7-11 to 7-13 show a comparison between photos taken during the mechanical testing of an OHT3 specimen and the numeric predictions for the OHT3 specimen with all the proposed models.

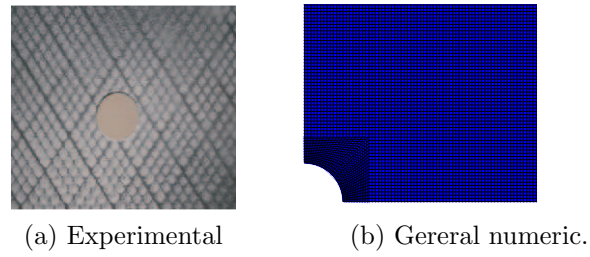


Figure 7-11: Illustration of Phase A in acoustic emission events: no damage.

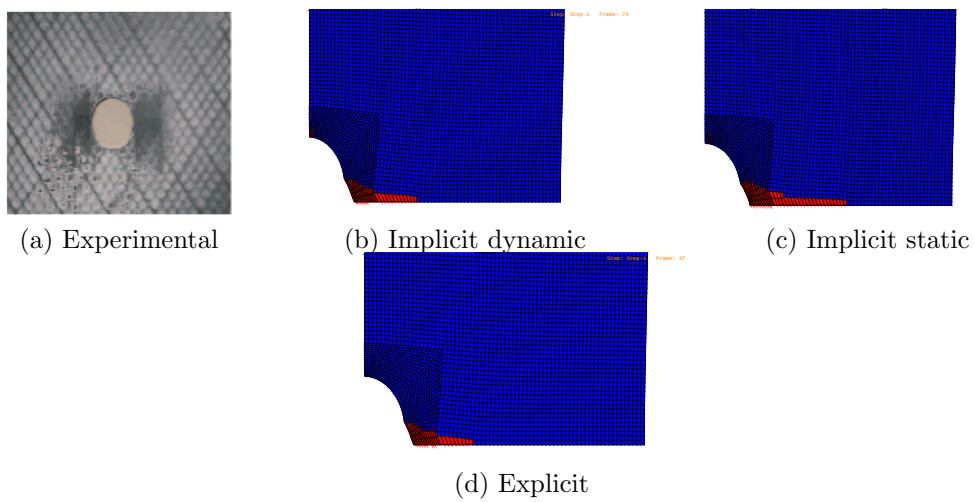


Figure 7-12: Illustration of Phase B in acoustic emission events: matrix cracking.

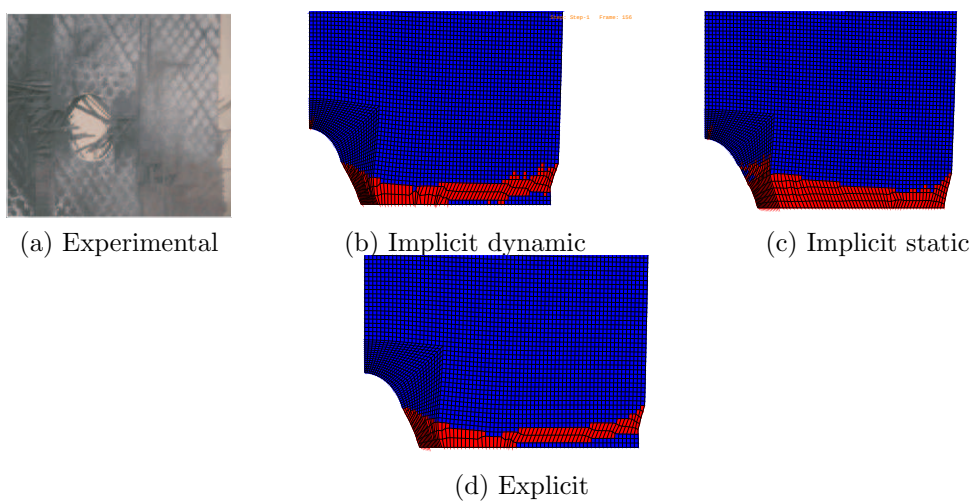


Figure 7-13: Illustration of Phase C in acoustic emission events: fibre cracking and ultimate failure.

The above Figures show the accuracy of the proposed continuum-damage model for damage initiation and evolution prediction. In the numeric models the distinction between matrix cracking prediction and fibre cracking prediction is made using a solution-dependent variable (SDV) for each type of failure. In Figure 7-12 the numeric prediction photos show a contour plot of SDV2, which corresponds to the SDV indicating matrix failure. In Figure 7-13 the numeric prediction photos show a contour plot of SDV1, which corresponds to the SDV indicating fibre failure. All the Implicit Dynamic, Implicit static and Explicit models captured correctly the damage initiation mechanism (matrix cracking) and the damage evolution (to fibre fracture) up to the specimen ultimate failure.

In Section 7.4 the predicted values for the specimens' ultimate failure loads are presented.

7.4 Ultimate failure load

In this Section, the ultimate failure loads predicted for each specimen simulated using the implicit dynamic, implicit static and explicit models are presented and compared with the experimental values.

The predicted ultimate failure loads are shown in Table 7.2 along side with the experimental ultimate failure loads. In Table 7.3 the relative error of the predictions is presented.

The results in Tables 7.2 and 7.3 for both the implicit models correlate well with the experimental results. The results for the implicit static model are consistently more accurate than for the implicit dynamic model. The absence of mass in the implicit static model can justify the better results obtained. The energy release peaks that occur when the first fibre fracture event is predicted are dissipated by the artificial damping included in the analysis and do not magnify the load transfer effect to the undamaged elements. In the implicit dynamic model the energy release peaks mentioned cause magnification of the load to be transferred to the undamaged elements and provoke a premature failure of these

Table 7.2: Ultimate failure loads measured and predicted.

Spec. Ref.	Experimental (N)	Impl. dynamic (N)	Impl. static (N)	Explicit (N)
OHT1	9196	7287	8490	13850
OHT2	26817	23260	26791	-
OHT3	48543	38935	41991	64984
OHT4	11997	9605	11008	15117
OHT5	34924	30200	34353	-
OHT6	54560	53294	53106	-
OHT7	15492	10928	13623	-
OHT8	41940	38949	42025	58697
OHT9	68200	64147	63615	75018

Table 7.3: Ultimate failure loads prediction error.

Spec. Ref.	Impl. dynamic (%)	Impl. static (%)	Explicit (%)
OHT1	-20.7	-7.6	+50.6
OHT2	-13.3	-0.1	-
OHT3	-19.8	-13.5	+33.9
OHT4	-19.9	-8.2	+26.0
OHT5	-13.5	-1.6	-
OHT6	-2.3	-2.7	-
OHT7	-29.5	-12.0	-
OHT8	-7.1	+0.2	+39.9
OHT9	-5.9	-6.7	+10.0

elements. If an artificial damping parameter was included in the implicit dynamic simulations, the results would improve.

For the explicit model the results do not correlate well with the experimental results and consistently over-predict the ultimate failure load value. The considerable computational cost of the simulations with the explicit model did not allow further assessment of the model. A possible explanation to the results obtained can be the "*mass-scaling*" operation performed to reduce the otherwise unbearable computational cost of these analysis. The "*mass-scaling*" operation dramatically increases the material density in order for the analysis to be com-

puted using a grater stable timestep. The effect of a greater inertia can cause the load transfer to undamaged elements to be less effective and therefore interfere with the predicted ultimate failure load value.

A remarkable result obtained using the model proposed is the simulation of deterministic size effects ("*hole-size effect*"). As shown in Table 7.4 the model predicts a reduction of strength when increasing the hole diameter in specimens with the same width-to-hole diameter ratio.

Table 7.4: Hole-size effect predictions with the implicit static model.

Spec. Ref.	d (mm)	w (mm)	w/d	σ^u (MPa)
OHT1	6	12	2	450.64
OHT4	8	16	2	438.22
OHT7	10	20	2	433.85
OHT2	6	24	4	474.01
OHT5	8	32	4	455.85
OHT8	10	40	4	446.13
OHT3	6	36	6	445.76
OHT6	8	48	6	422.82
OHT9	10	60	6	405.19

Figure 7-14 shows the predicted effect using the implicit static model results.

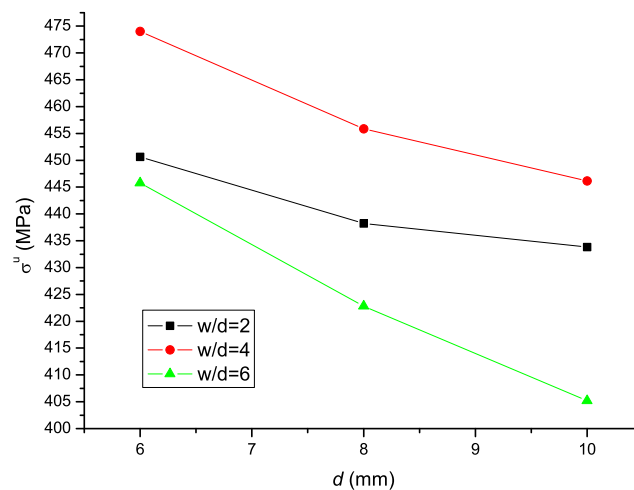


Figure 7-14: Hole-size effect predictions with the implicit static model.

Chapter 8

Conclusions

The Continuum-Damage model proposed in this work has proven to be an effective and accurate tool to predict damage initiation and damage evolution in advanced composite materials.

It is important to highlight the difficulty of predicting the behaviour of structures made from advanced composite materials. For a correct simulation of the behaviour of this type of material, every constituent has to be analyzed independently and then the constituent interaction must be correctly computed. In order to capture the effect of all the interactions in the material, any proposed simulation model will always be computational cost intensive. The computational cost also adds to the difficulty of developing such models as it takes a considerable amount of time to obtain the results from any modification to the computational code. Besides taking into account the above mentioned material behaviour characteristics, stress concentration factors such as holes and boundary proximity effects are also contemplated in this work adding to the overall difficulty of the predictions presented.

All the proposed approaches have correctly captured the damage initiation mechanism for the tested specimens and have made accurate predictions for the damage evolution to other failure mechanisms.

For the implicit static model the results proved to correlate extremely well with the experimental results from the onset of damage to ultimate failure load

prediction.

An important conclusion of this work is that the integration-point discount method, based on a sudden reduction of the elastic properties, should be used in implicit static analysis. When using implicit dynamic analysis the stress waves generated by the sudden release of elastic energy result in premature damage of the material in the vicinity of the damaged region.

The explicit model has also correctly captured the damage initiation mechanism and the damage evolution for the tested specimens. However the results for the ultimate failure load predictions were considerably worst than the results obtained with the implicit models. Due to the high computational cost of the simulations using the explicit model, it was not possible to further assess the problems with this model in the duration of this work. A possible cause for the offset in the predicted values may be related to the "*mass-scaling*" operation performed to reduce the computational cost of these simulations and needs to be investigated in more detail.

There is still work ahead for improvement to the proposed Continuum-Damage model. The fundamental improvement to be made has to be the material property degradation scheme. The degradation scheme used implements an abrupt mechanical property reduction which spans element-wise. This approach clearly produces mesh-dependent results as the energy release peaks that occur when damage is predicted within an element are a function of the element size. A more suitable mechanical property degradation scheme would progressively degrade the material mechanical properties within an element to more accurately simulate the load transfer effects to the undamaged elements.

The objective of developing a tool to aid the design process of structures with advanced composite materials has been accomplished. The proposed Continuum-Damage model can be used to predict the mechanical behaviour of advanced composite structures and can be a very useful tool in fine-tuning a conceptual structure for the construction of a test prototype and therefore being a potentially important engineering and economical asset.

Bibliography

- [1] Hibbitt, Karlsson and Sorensen. 2002. ABAQUS 6.3 User's Manuals. Pawtucket, U.S.A.
- [2] Whitney, J. M. and Nuismer, R. J. Stress Fracture Criteria for Laminated Composites Containing Stress Concentrations. Journal of Composite Materials, Vol.8, pp.253-265, 1974.
- [3] Waddoups, M. E.; Eisenmann, J. R., and Kaminski, B. E. Macroscopic fracture mechanics of advanced composite materials. Journal of Composite Materials, Vol.5, pp.446-454, 1971.
- [4] Ladevèze, P. Sur une Théorie de L'Endommagement Anisotrope. Rapport Interne 34. Laboratoire de Mécanique et Technologie. Cachan, Fr., 1983.
- [5] Ladevèze, P.; Allix, O.; Deu, J.-F., and Lévêque, D. A Mesomodel for Localisation and Damage Computation in Laminates. Computer Methods in Applied Mechanics and Engineering, Vol.183, pp.105-122, 2000.
- [6] Ladevèze, P.; Guitard, L.; Champaney, L., and Aubard, X. Debond Modelling for Multidirectional Composites. Computer Methods in Applied Mechanics and Engineering, Vol.185, pp.109-122, 2000.
- [7] Ladevèze, P. and Lubineau, G. On a Damage Mesomodel for Laminates: Micro-Meso Relationships, Possibilities and Limits. Composites Science and Technology, Vol.61, pp.2149-2158, 2001.

- [8] Ladevèze, P. and Lubineau, G. An enhanced Mesomodel for Laminates based on Micromechanics. *Composites Science and Technology*, Vol.62, pp.533-541, 2002.
- [9] Ladevèze, P. and Lubineau, G. On a Damage Mesomodel for Laminates: Micromechanics Basis and Improvements. *Mechanics of Materials*, Vol.35, pp.763-775, 2003.
- [10] Ladevèze, P. and Lubineau, G. Pont Entre Les *Micro* et *Méso* Mécaniques Des Composites Stratifiés. *Comptes Rendus Mecanique*, Vol.331, pp.537-544, 2003.
- [11] Ladevèze, P. Multiscale Modelling and Computational Strategies for Composites. *International Journal for Numerical Methods in Engineering*, Vol.60, pp.233-253, 2004.
- [12] "Standard Test Methods for Constituent Content of Composite Materials", ASTM D 3171-99, American Society for Testing and Materials (ASTM), West Conshohocken, PA, U.S.A.
- [13] The Mathworks, Inc. Matlab 7.0 User's Manuals. Natick, MA, U.S.A.
- [14] "Standard test method for tensile properties of polymer matrix composite materials", ASTM D 3039/D 3039M-00, American Society for Testing and Materials (ASTM), West Conshohocken, PA, U.S.A.
- [15] "Standard Test Method for Compressive Properties of Unidirectional or Cross-ply Fiber-Resin Composites", ASTM D 3410-87., American Society for Testing and Materials (ASTM), West Conshohocken, PA, U.S.A.
- [16] "Standard test method for in-plane shear response of polymer matrix composite materials by test of a ± 45 laminate" ASTM D 3518/3518M-94, American Society for Testing and Materials (ASTM), West Conshohocken, PA, U.S.A.

- [17] P.P. Camanho, C.G. Dávila, S.T. Pinho, L. Iannucci, Robinson, P. "Prediction of in-situ strengths and matrix cracking in composites under transverse tension and in-plane shear", submitted, Composites-Part A, 2005.
- [18] S.T. Pinho, C.G. Dávila, P.P. Camanho, L. Iannucci, P. Robinson, "Failure models and criteria for FRP under in-plane or three-dimensional stress states including shear non-linearity", NASA Technical Memorandum 213530, National Aeronautics and Space Administration, U.S.A., 2005.
- [19] "Standard Test Methods for Mode I Interlaminar Fracture Toughness of Unidirectional Fiber-Reinforced Polymer Matrix Composites", ASTM D 5528-01, American Society for Testing and Materials (ASTM), West Conshohocken, PA, U.S.A.
- [20] C.G. Dávila, P.P. Camanho, L. Iannucci, P. Robinson, "Failure criteria for FRP in plane stress", NASA Technical Memorandum 212663, National Aeronautics and Space Administration, U.S.A., 2003.
- [21] Hahn, H. T. and Tsai, S. W. Nonlinear elastic behaviour of unidirectional composite laminate. *Journal of Composite Materials*. 1973; 7:102-110.
- [22] "Open hole tensile strength of polymer composite laminates", ASTM D 5766/D 5766M-02a, American Society for Testing and Materials (ASTM), West Conshohocken, PA, U.S.A.
- [23] Pollock A.A. Acoustic emission inspection, *Non Destructive Evaluation and Quality Control*. Metals Handbook, Vol. 17, ASM International, pp.278-294, 1989.