



Politecnico di Torino & ISAE-SUPAERO

Master Degree Thesis in Aerospace Engineering

Academic Year 2025–2026

Damage and failure identification of composite laminates with carbon fibers and thermoset matrix under static and dynamic loadings

Supervisors:

Prof. Marco Gherlone
Prof. Frédéric Lachaud

Candidate:

Roberto Ostuni

Abstract

Carbon fiber reinforced polymer matrix composites are increasingly used in the aeronautical sector due to their high strength-to-weight ratio and the possibility of tailoring the mechanical properties of the laminate. Among the main damage mechanisms, delamination represents a critical phenomenon that can significantly compromise the structural integrity of components.

The present thesis work, carried out during an internship at ISAE-Supaero in collaboration with Airbus Atlantic and Institut Clément Ader, is aimed at the experimental and numerical characterization of the main mechanical properties of a thermosetting polymer composite reinforced with carbon fibers, with particular attention to interlaminar fracture toughness in Mode I and Mode II.

The experimental campaign was conducted in accordance with ASTM standards. In particular, three-point bending tests were performed for the determination of the flexural modulus, Double Cantilever Beam (DCB) tests were carried out for Mode I fracture characterization, and both End-Notched Flexure (ENF) and Four-Point End-Notched Flexure (4ENF) configurations were employed for Mode II characterization.

The fracture toughness parameters were evaluated using different approaches, combining energy-based formulations with compliance-based methods.

In order to validate the experimental findings, numerical models were subsequently developed in Abaqus. Fracture behavior was numerically simulated using two principal modeling strategies: the Virtual Crack Closure Technique (VCCT) and the Cohesive Zone Model (CZM).

Finally, the response of the material under dynamic loading conditions is preliminarily addressed in order to explore strain-rate effects. Initial activities related to Split Hopkinson Pressure Bar (SHPB) testing are carried out, constituting the starting point for further investigations that will be pursued in subsequent research.

Contents

Abstract	1
1 Introduction	1
1.1 Context and Motivation	1
1.2 Objectives of the Work	2
1.3 Structure of the Thesis	2
2 Theoretical Background	4
2.1 Composite Materials	4
2.1.1 Definition of Composite Material	4
2.1.2 Classification of Composite Materials	4
2.1.3 Fiber-Reinforced Composites	6
2.1.4 Laminated Composites	6
2.2 Defects and Damage Mechanisms in Composite Materials . .	7
2.2.1 Interfacial Debonding	7
2.2.2 Matrix Microcracking	7
2.2.3 Fiber Breakage	8
2.2.4 Delamination	8
2.3 Fracture Modes	8
2.4 Interlaminar fracture toughness	9
2.4.1 Griffith's Theory	9
2.4.2 Compliance method	10
2.5 Tests	10
2.5.1 Three-Point Bending Test	10
2.5.2 Double Cantilever Beam (DCB) – Test Theory	12
2.5.3 End-Notched Flexure (ENF) – Test Theory	14
2.5.4 Four-Point End-Notched Flexure (4ENF)	16
2.6 Numerical modeling of delamination	19
2.6.1 Virtual Crack Closure Technique (VCCT)	19
2.6.2 Cohesive Zone Model (CZM)	20

3	Experimental Tests	22
3.1	Material and Experimental Equipment	22
3.1.1	Material	22
3.1.2	Testing Machine	23
3.2	Three-Point Bending (3PB) — Flexural Modulus Identification	24
3.2.1	Specimen and Experimental Setup	24
3.2.2	Data Reduction and Flexural Modulus Evaluation . .	25
3.2.3	Results and Discussion	26
3.3	Tensile Test ($\pm 60^\circ$) — In-Plane Elastic Properties	28
3.3.1	Specimen and Experimental Setup	28
3.3.2	Data Reduction and Identification of Elastic Parameters	30
3.3.3	Results and Discussion	31
3.4	Double Cantilever Beam (DCB) — Mode I Interlaminar Frac- ture Toughness	35
3.4.1	Specimen and Experimental Setup	35
3.4.2	Data Reduction and Fracture Toughness Evaluation .	37
3.4.3	Results and Discussion	38
3.5	End-Notched Flexure (ENF) — Mode II Interlaminar Frac- ture Toughness	42
3.5.1	Specimen and Experimental Setup	42
3.5.2	Data Reduction and Evaluation of G_{IIc}	43
3.5.3	Results and Discussion	44
3.6	Four-Point End-Notched Flexure (4ENF) — Mode II Inter- laminar Fracture Toughness	48
3.6.1	Specimen and Experimental Setup	48
3.6.2	Data Reduction and Evaluation of G_{IIc}	49
3.6.3	Results and Discussion	50
3.7	Discussion of Experimental Results	52
4	Numerical Modelling Results	53
4.1	Interlaminar Fracture Modelling Approach	54
4.1.1	Virtual Crack Closure Technique (VCCT)	54
4.1.2	Surface-Based Cohesive Contact	54
4.2	Three-Point Bending (3PB)	55
4.2.1	Numerical Setup	55
4.2.2	Results and Comparison	57
4.3	Tensile Test ($\pm 60^\circ$)	61
4.3.1	Numerical Setup	61
4.3.2	Results and Comparison	65
4.4	Double Cantilever Beam (DCB)	70
4.4.1	Numerical Setup	70
4.4.2	Results and Comparison	72
4.5	End-Notched Flexure (ENF)	77
4.5.1	Numerical Setup	77

4.5.2	Results and Comparison	79
4.6	Four-Point End-Notched Flexure (4ENF)	85
4.6.1	Numerical Setup	85
4.6.2	Results and Comparison	88
4.7	Discussion of Numerical Results	93
5	High Strain Rate Behaviour of Composite Materials	95
5.1	Introduction	95
5.2	Strain Rate Effects in CFRP Laminates	96
5.2.1	Longitudinal Tensile Behaviour of Unidirectional Lam- inates (0°)	96
5.2.2	Off-Axis Tensile Behaviour (45° and 90°)	97
5.2.3	Cross-Ply Laminates	97
5.2.4	Longitudinal Compressive Behaviour of Unidirectional Laminates (0°)	98
5.3	Split Hopkinson Pressure Bar Technique	99
5.3.1	Stress-Strain Determination in SHPB Tests	100
6	Conclusions	102
A	Additional Manufacturing Attempts	104
A.1	Manufacturing of an L-shaped specimen for through-thickness characterization	104
A.1.1	Specimen preparation	104
A.1.2	Vacuum bagging configuration	105
A.1.3	Schematic representation of the bagging stack	107
A.1.4	Temperature monitoring	107
A.1.5	Curing issue and testing limitation	108

List of Figures

2.1	Classification scheme of composite materials based on the matrix constituent material and the type of reinforcement. . . .	5
2.2	Experimental setup of the three-point bending (3PB) test: (a) real testing configuration; (b) geometric scheme of the specimen.	11
2.3	Experimental setup of the Double Cantilever Beam (DCB) test.	12
2.4	Force–displacement curve; triangular areas represent the energy variation ΔU used to compute G_{Ic}	14
2.5	End-Notched Flexure (ENF) test configuration: (a) real experimental setup; (b) geometrical scheme.	14
2.6	Experimental response of the ENF test and construction of the calibration relationship for the calculation of G_{IIc} using the compliance method: (a) force–displacement curves as a function of a_0 ; (b) calibration law $C(a^3)$	16
2.7	Four-Point End-Notched Flexure (4ENF) test: (a) experimental setup; (b) geometrical configuration and dimensional parameters.	17
2.8	Force–displacement curve of the 4ENF test.	18
2.9	Two-dimensional scheme of the VCCT method for four-node quadrilateral elements.	20
2.10	Bilinear traction–separation law of the cohesive model (CZM).	21
3.1	Composite material supplied by Airbus Atlantic: cured laminate panels and prepreg roll used for specimen manufacturing.	23
3.2	INSTRON 5969 testing machine.	24
3.3	Geometrical dimensions of the three-point bending specimen.	25
3.4	Force–displacement curve highlighting the 10%–50% F_{peak} interval used for the calculation of the flexural modulus.	26
3.5	Force–displacement curves obtained from the three-point bending tests.	27
3.6	Geometry of the $\pm 60^\circ$ tensile specimen.	28
3.7	Reference dots used for the optical measurement of axial and transverse strains.	29

3.8	Optical acquisition system used to track the displacement of the reference dots during the tensile test.	30
3.9	Force–displacement curves obtained from the tensile tests. . .	31
3.10	Stress–strain behaviour in the longitudinal (x) and transverse (y) directions.	32
3.11	Evolution of Poisson’s ratio ν_{xy} during the tensile tests as a function of strain and stress.	33
3.12	Geometrical scheme of the Double Cantilever Beam specimen	36
3.13	Microscope used to verify initial crack length	37
3.14	Force–displacement curves for the five DCB tests.	38
3.15	Evolution of G_{Ic} as a function of opening displacement. . . .	39
3.16	Evolution of G_{Ic} calculated using inclinometers.	39
3.17	Fracture surfaces of the Test 1 of DCB specimens.	40
3.18	Fracture surface of the DCB specimen Test 3.	42
3.19	Geometrical scheme of the End-Notched Flexure.	43
3.20	ENF:Force vs Displacement.	44
3.22	Geometrical scheme of the 4End-Notched Flexure.	49
3.23	4ENF:Force vs Displacement.	50
4.1	Finite element model of the three-point bending specimen. . .	55
4.2	Boundary conditions and loading configuration of the three-point bending model.	56
4.3	Finite element mesh of the three-point bending model.	57
4.4	Deformed configuration of the three-point bending specimen (vertical displacement U_2).	58
4.5	Distribution of longitudinal normal stress σ_{11} in the three-point bending specimen.	59
4.6	Distribution of transverse normal stress σ_{22}	60
4.7	Comparison between experimental and numerical force–displacement curves for the three-point bending test.	60
4.8	Finite element model of the tensile specimen.	62
4.9	Composite layup definition of the tensile specimen.	62
4.10	Numerical setup of the tensile test model: boundary conditions and interaction definition.	64
4.11	Finite element mesh of the tensile specimen.	65
4.12	In-plane displacement fields of the tensile specimen.	66
4.13	Stress distribution in the tensile specimen.	68
4.14	Comparison between experimental and numerical force–displacement curves for the tensile test.	69
4.15	Finite element model of the DCB specimen.	70
4.16	Numerical setup of the DCB model: boundary conditions and interfacial interaction definitions.	71
4.17	Finite element mesh of the DCB model.	72
4.18	Deformed configuration of the DCB model.	73

4.19	Distribution of σ_{22} at peak load.	73
4.20	$G_{Ic} = 0.25$ N/mm: comparison between experimental and numerical force–displacement curves, including a zoom of the peak region.	74
4.21	$G_{Ic} = 0.35$ N/mm: comparison between experimental and numerical force–displacement curves, including a zoom of the peak region.	75
4.22	$G_{Ic} = 0.45$ N/mm: comparison between experimental and numerical force–displacement curves, including a zoom of the peak region.	75
4.23	Finite element model of the ENF specimen.	77
4.24	Boundary conditions and loading configuration of the ENF model.	78
4.25	Finite element mesh of the ENF model.	79
4.26	Deformed configuration of the ENF model (U_2 displacement field).	80
4.27	Distribution of σ_{12} at peak load for the ENF model.	80
4.28	Distribution of σ_{22} in the ENF model.	81
4.29	Comparison between experimental and numerical ENF force–displacement curves for different initial crack lengths a_0 (Tests 1 and 2).	82
4.30	Comparison between experimental and numerical ENF force–displacement curves for different initial crack lengths a_0 (Tests 3 and 4).	83
4.31	Comparison between experimental and numerical ENF force–displacement curves for different initial crack lengths a_0 (Test 5).	84
4.32	Finite element model of the 4ENF specimen.	86
4.33	Numerical setup of the 4ENF model: boundary conditions and interfacial interaction definitions.	87
4.34	Finite element mesh of the 4ENF model.	88
4.35	Deformed configuration of the 4ENF specimen under loading (U_2 displacement field).	89
4.36	Shear stress distribution σ_{12} at peak load in the 4ENF specimen.	90
4.37	Transverse normal stress distribution σ_{22} in the 4ENF specimen.	91
4.38	Comparison between experimental and numerical force–displacement curves for Test 1.	92
4.39	Comparison between experimental and numerical force–displacement curves for Tests 2 and 3.	93
5.1	Schematic representation of a Split Hopkinson Pressure Bar (SHPB).	99

A.1	Layup of the carbon/epoxy prepreg laminate on the L-shaped metallic tool.	105
A.2	Vacuum bagging configuration prepared before the curing cycle.	106
A.3	Schematic representation of the vacuum bagging stack used during the manufacturing process.	107
A.4	Positioning of thermocouples used to monitor the temperature during the curing cycle.	108
A.5	Demoulded L-shaped composite specimen after the curing process.	109

List of Tables

3.1	3PB dimensions	25
3.2	Experimental results of the three-point bending tests.	27
3.3	Geometrical parameters of the specimen.	28
3.4	Summary of the Poisson's ratio ν_{xy} and longitudinal Young's modulus E_x obtained from the tensile $\pm 60^\circ$ tests.	34
3.5	DCB dimensions	36
3.6	G_{IC} Results	40
3.7	ENF dimensions	43
3.8	ENF test results	47
3.9	4ENF dimensions	49
3.10	4ENF experimental results.	51
4.1	Engineering constants used for the numerical simulations.	53

Chapter 1

Introduction

1.1 Context and Motivation

Composite materials are increasingly used in the aeronautical industry due to their excellent strength-to-weight ratio compared to traditional metallic materials. Their use makes it possible to significantly reduce the structural mass of aircraft while maintaining high mechanical performance, with direct benefits in terms of efficiency, fuel consumption, and overall operational performance.

In the aeronautical field, the most commonly used reinforcements are glass fibers and carbon fibers. Among them, carbon fibers are widely employed for high-performance structural applications thanks to their high specific stiffness and strength. As far as the polymer matrix is concerned, a distinction can mainly be made between thermosetting and thermoplastic matrices.

Carbon fibers are often combined with thermosetting epoxy matrices, since this combination provides high stiffness and strength while maintaining relatively reasonable industrial costs. However, thermosetting matrices also present some intrinsic limitations. Their brittle nature results in a relatively low resistance to crack initiation and propagation. In addition, they may absorb moisture, which can affect mechanical properties, and their manufacturing process generally requires complex multi-step curing cycles.

Thermoplastic matrices represent a promising alternative from a technological point of view. They generally exhibit higher toughness and good resistance to elevated temperatures due to their higher glass transition temperature. They also show good resistance to corrosive environments and, unlike thermosetting matrices, they can be recycled. Despite these advantages, thermoplastic composites are still less widely used in aeronautical structures, mainly because of the technological challenges related to their processing and consolidation techniques, which are still the subject of ongoing research and development. [1]

In this context, a detailed mechanical characterization of composite materials currently used in aeronautical structures becomes essential. In particular, understanding damage mechanisms and fracture behaviour is crucial, since these phenomena strongly influence the structural safety and long-term reliability of composite components.

The work presented in this thesis was carried out in collaboration with ISAE-SUPAERO, the Institut Clément Ader, and Airbus Atlantic. The study focuses on the mechanical characterization of a thermosetting carbon fiber reinforced composite with an epoxy matrix, representative of materials widely used in aeronautical structures. Particular attention is devoted to the study of crack initiation and propagation, with a specific focus on interlaminar delamination phenomena.

1.2 Objectives of the Work

The main objective of this work is the mechanical characterization of a carbon fiber reinforced thermosetting composite material through an experimental campaign performed under laboratory conditions, followed by the validation of the obtained results through comparison with numerical simulations.

More specifically, the material was subjected to tensile tests and three-point bending tests in order to determine its global mechanical properties. In addition, fracture mechanics tests such as Double Cantilever Beam (DCB), End Notched Flexure (ENF), and Four-Point End Notched Flexure (4ENF) tests were carried out to investigate interlaminar fracture toughness and the mechanisms governing delamination propagation under Mode I and Mode II loading conditions.

The experimental results were then compared with numerical simulations developed using the Abaqus software. This comparison makes it possible to evaluate the capability of the numerical models to reproduce the experimentally observed mechanical behaviour and to support the interpretation of the fracture mechanisms involved.

1.3 Structure of the Thesis

The thesis is organized as follows.

Chapter 2 presents the theoretical background necessary to understand the behaviour of composite materials, including their classification, the main concepts related to fracture toughness, and the fundamental mechanisms governing damage and crack propagation. This chapter also introduces the theoretical principles underlying the experimental tests and the numerical modelling approaches adopted in this work.

Chapter 3 presents the results obtained from the experimental testing campaign, describing the experimental setups and the procedures followed for each test.

Chapter 4 describes the numerical models developed using Abaqus and discusses the results of the simulations, which are compared with the experimental findings.

Chapter 5 provides a brief overview of dynamic testing techniques and discusses the influence of strain rate on the mechanical response of composite materials. In particular, the Split Hopkinson Pressure Bar (SHPB) technique is introduced from a theoretical point of view, describing its working principles and highlighting its potential use for future strain-rate-dependent characterisation.

Finally, Chapter 6 summarizes the main conclusions of the work and outlines possible directions for future developments.

Chapter 2

Theoretical Background

2.1 Composite Materials

2.1.1 Definition of Composite Material

Composite materials are continuously evolving and are experiencing increasing use in numerous engineering fields, due to the possibility of obtaining combinations of mechanical and functional properties that are difficult to achieve with traditional homogeneous materials.

In general, a composite material is formed by the combination of two or more constituents with different physical and chemical properties which, when properly combined, give rise to a final material with overall characteristics that are different and often superior to those of the individual components [2].

In general terms, composite materials consist of two main phases: the reinforcement, typically in the form of fibers, which provides the material with its main stiffness and mechanical strength properties, and the matrix, which acts as a structural binder, ensuring material continuity and transferring the applied loads to the fibers and, consequently, to the entire composite [3].

2.1.2 Classification of Composite Materials

Composite materials exhibit high variability in terms of constituents, configurations, and combination methods. For this reason, it is essential to introduce appropriate classification criteria that allow different types of composites to be organized and distinguished according to their main characteristics and application fields.

An overview of the main classification criteria of composite materials is shown in Figure 2.1, adapted from [3].

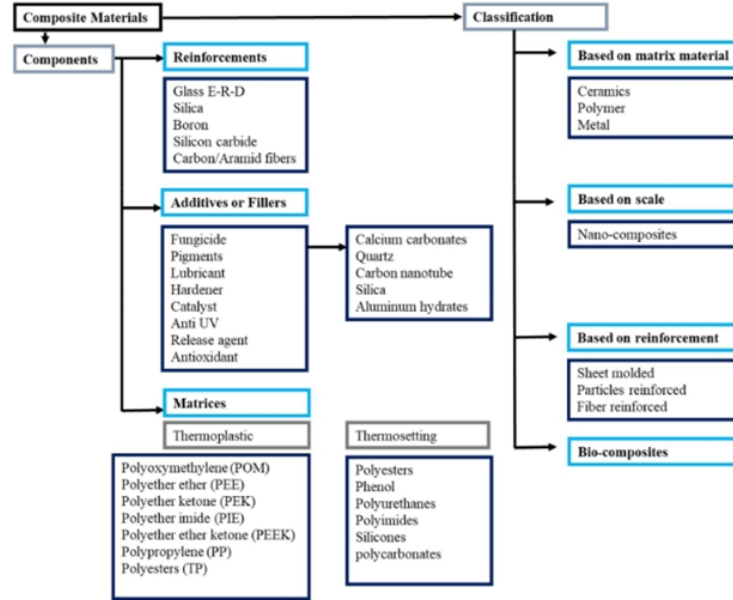


Figure 2.1: Classification scheme of composite materials based on the matrix constituent material and the type of reinforcement, adapted from [3].

In particular, composites can be distinguished according to the matrix material and the type of reinforcement used.

Based on the reinforcement type, composite materials can be divided into particle-reinforced, sheet-reinforced, and fiber-reinforced composites. Among these, fiber-reinforced composites represent the most relevant solution for high-performance structural applications, due to the high stiffness and strength achievable along the fiber orientation directions [2,3].

In the present case, a thermosetting polymer matrix composite reinforced with carbon fibers is considered. Due to the preferential orientation of the fibers, the material exhibits anisotropic mechanical behavior, which significantly influences its response under mechanical loading and its damage mechanisms.

2.1.3 Fiber-Reinforced Composites

Fiber-reinforced composite materials can exhibit different structures depending on fiber length, arrangement, and orientation. In particular, fibers can be short or long: in the first case, the reinforcement is distributed more randomly within the matrix, whereas in the second case a more continuous structure is obtained, typically associated with improved mechanical performance.

Fibers may also be arranged in unidirectional or bidirectional configurations. Fiber orientation is one of the fundamental aspects determining the mechanical properties of the composite, since the fiber direction generally coincides with the direction of maximum stiffness and strength of the material.

Consequently, the mechanical behavior of the composite is anisotropic, with properties strongly dependent on the direction of the applied load. In the present work, different fiber orientation configurations are considered, depending on the type of experimental test performed. In general, the analyzed material consists of a laminate with fibers oriented along predefined directions, which is typical of structural applications in the aeronautical field [4].

2.1.4 Laminated Composites

Laminated composites are structures composed of a stacking sequence of layers (*plies*), made of identical or different materials and characterized by the same or different orientations. This architecture allows the material to be designed in a targeted way, adapting its mechanical properties to specific structural requirements; for this reason, laminated composites are widely used in aeronautical engineering.

However, such materials also present important critical aspects from both manufacturing and structural perspectives. The presence of interfaces between layers with different orientations represents a potential site for defects, such as interlaminar discontinuities and delamination phenomena, which may compromise the mechanical performance of the laminate and promote crack initiation and propagation [5].

To limit these undesired effects, stacking sequences are generally designed to be symmetric with respect to the laminate mid-plane. Symmetry prevents the occurrence of coupling between extension and bending deformations, simplifying the global mechanical behavior of the material and reducing sensitivity to defects introduced during the manufacturing process.

2.2 Defects and Damage Mechanisms in Composite Materials

Although composite materials exhibit numerous advantages and excellent mechanical properties, their heterogeneous structure, the marked difference between constituents, and the presence of interfaces between fibers, matrix, and laminate layers represent potential sites for damage initiation. Moreover, the anisotropy introduced by the directional nature of the reinforcements contributes to making the mechanical behavior of the material complex and strongly dependent on the applied loading direction.

Whenever an interface is present within the composite, such as the fiber–matrix interface or the interface between adjacent laminate plies, the probability of defect formation under mechanical loading increases. In general, several damage mechanisms may progressively degrade the mechanical properties of the material. For this reason, it is essential to characterize the behavior of the composite in the presence of damage, in order to prevent structural collapse phenomena or a significant degradation of performance. This section describes the main defects observed in fiber-reinforced composite materials [6].

2.2.1 Interfacial Debonding

The performance of a fiber-reinforced composite strongly depends on the quality of the bond at the fiber–matrix interface, since this interface governs load transfer from the matrix to the reinforcements. In the presence of insufficient adhesion, even low levels of applied stress may lead to crack formation along the fiber–matrix interface.

The *debonding* phenomenon may also be promoted by the different mechanical response of the two constituents: under loading, matrix and fibers undergo different deformations, locally generating stresses due to the Poisson effect. When these stresses exceed the interfacial strength, fiber detachment from the matrix occurs [6].

2.2.2 Matrix Microcracking

Fiber-reinforced composite materials exhibit high stiffness and strength along the fiber direction; however, these properties are significantly lower in the transverse direction. This marked anisotropy may lead to the formation of microcracks within the matrix, typically oriented parallel to the fibers.

Although such microcracks do not generally cause immediate structural failure, they lead to a degradation of the material mechanical properties and may promote the initiation of additional damage mechanisms [6].

2.2.3 Fiber Breakage

In a unidirectional composite loaded along the fiber direction, individual fiber failure typically occurs at their weakest points. This event results in a redistribution of stresses between adjacent fibers and the matrix, causing a local increase in the stress level.

The fiber failure process has a statistical nature but is strongly influenced by the presence of local defects and by the efficiency of the fiber–matrix bond, which governs the ability of the material to redistribute stresses [6].

2.2.4 Delamination

Delamination is an interlaminar defect consisting of the separation between two adjacent plies of a laminate. It frequently occurs near free edges, holes, or geometric discontinuities. Under in-plane loading, the composite develops interlaminar shear stresses through the thickness, which may initiate and propagate delamination.

The propagation of delamination leads to a significant reduction in the mechanical properties of the laminate. The resistance to delamination initiation is mainly related to the properties of the matrix, whereas crack growth is governed by the interlaminar fracture toughness of the material. Delamination represents one of the main damage mechanisms investigated in the present experimental work [6].

2.3 Fracture Modes

In the present thesis, the focus is mainly on the study of delamination in composite materials. The propagation of this type of damage may occur according to different modes, which are classified based on the stress state acting at the crack front. In fracture mechanics, these modes are commonly identified as Mode I, Mode II, and Mode III.

Mode I, also referred to as the *opening mode*, occurs when the delamination is subjected to a tensile load perpendicular to the crack plane. Under this condition, the delaminated surfaces tend to separate from each other, promoting crack opening and damage propagation.

Mode II, known as the *in-plane sliding mode*, is characterized by the relative sliding of the delaminated surfaces due to a shear stress state applied within the plane of the laminate. This type of loading induces crack propagation mainly governed by interlaminar shear mechanisms.

Mode III, or *out-of-plane sliding mode*, is associated with a shear stress state applied in a direction orthogonal to the laminate plane, causing relative sliding of the crack surfaces along the thickness direction.

Due to the anisotropic and heterogeneous nature of composite materials, multiple fracture modes may simultaneously be present at the delamination

front, resulting in mixed-mode loading conditions [7].

Within the context of the present thesis, the analysis is exclusively focused on Mode I and Mode II fracture, whereas Mode III is not considered, as it is not investigated experimentally and is not relevant to the objectives of this study.

2.4 Interlaminar fracture toughness

Composite materials are particularly susceptible to interlaminar failure, especially in the case of continuous carbon or glass fibers embedded in a brittle polymer matrix. In such materials, the presence of interfaces between adjacent plies makes delamination one of the main structural damage mechanisms.

The interlaminar fracture toughness of a composite is described through the interlaminar fracture energy release rate, denoted as G . This quantity represents the energy available for delamination propagation and is commonly analyzed within the framework of Linear Elastic Fracture Mechanics (LEFM) [8].

2.4.1 Griffith's Theory

Crack propagation can be described in energetic terms by introducing the concept of energy release rate G . This quantity represents the variation of the total potential energy of the system per unit area of newly created crack surface and is defined as

$$G = -\frac{d\Pi}{dA}, \quad (2.1)$$

where Π is the total potential energy of the system and A is the crack area.

In the energetic model introduced by Griffith, crack propagation is governed by a balance between the elastic energy released by the system and the energy required to create new fracture surfaces. Crack growth occurs when the released energy is sufficient to compensate for the increase in surface energy associated with crack advancement.

Based on this approach, it is possible to introduce a critical value of the energy release rate, denoted as G_C , which represents a characteristic property of the material and of the considered interface. The parameter G_C expresses the minimum energy required for crack propagation. When $G < G_C$, the crack remains stable, whereas for $G \geq G_C$, propagation becomes energetically possible, either in a stable or unstable manner depending on the loading conditions and structural configuration [9].

In laminated composite materials, and particularly in the case of delamination phenomena, the value of G_C is strongly influenced by the properties of the matrix and of the interlaminar interface. For this reason, the experimental determination of the interlaminar fracture toughness represents a

fundamental aspect for assessing the structural integrity of the material and constitutes the focus of the experimental tests presented in the following chapters.

2.4.2 Compliance method

The compliance method is an experimental approach used for the determination of the critical fracture energy G_C . It is based on the definition of compliance C , expressed as

$$C = \frac{\delta}{F}, \quad (2.2)$$

where δ represents the displacement applied to the specimen and F the corresponding load. Compliance can be interpreted as the inverse of the global stiffness of the system.

Compliance depends on the crack length; therefore, in order to correctly apply this method, it is essential to know the initial crack length and its evolution during the test. As crack propagation proceeds, the corresponding compliance value is calculated for each increment of crack growth.

The critical fracture energy can then be calculated using the relation

$$G_C = \frac{F^2}{2B} \frac{dC}{da}, \quad (2.3)$$

where B represents the specimen width and a the crack length.

In the following sections, the procedure used to determine compliance for each experimental test will be illustrated. In particular, for Mode II, the compliance curve is not obtained from a single test, but from multiple tests performed on specimens characterized by different initial crack lengths a_0 , in order to construct a single compliance curve as a function of a_0 .

2.5 Tests

In this section, the experimental tests performed within the framework of this work are described. For each test, an essential theoretical background is provided in order to support the subsequent presentation and interpretation of the experimental results, with particular attention to the procedures adopted for the calculation of the mechanical quantities of interest. The data processing methodologies are based on the principles introduced in Section 2.4.

2.5.1 Three-Point Bending Test

The Three-Point Bending (3PB) test, performed according to ASTM D790, is used for the determination of the flexural elastic modulus of the composite material. The test consists of bending a rectangular cross-section specimen

supported on two lower supports and loaded through a central indenter, as illustrated in Figure 4.2.

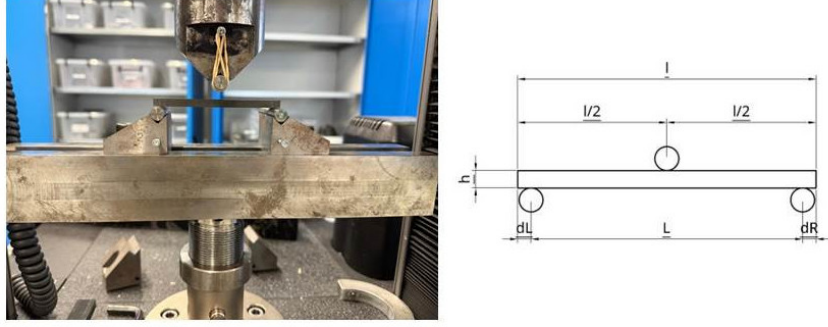


Figure 2.2: Experimental setup of the three-point bending (3PB) test: (a) real testing configuration; (b) geometric scheme of the specimen.

During the test, the specimen is subjected to a bending stress state that generates tensile stresses on one face and compressive stresses on the opposite face, with an intermediate neutral surface. Assuming linear elastic behavior and modeling the specimen as an equivalent homogeneous beam, the vertical displacement δ at the loading point is described, according to Euler–Bernoulli beam theory, by the relation:

$$\delta = \frac{FL^3}{48E_f I_z} \quad (2.4)$$

where F is the applied force, L is the span between the lower supports, E_f is the flexural modulus of the laminate, and I_z is the second moment of area of the cross-section, expressed as:

$$I_z = \frac{Bh^3}{12} \quad (2.5)$$

with B the specimen width and h the total laminate thickness.

From the initial slope of the force–displacement curve, the flexural modulus can be obtained as:

$$E_f = \frac{\Delta F}{\Delta \delta} \frac{L^3}{48I_z} \quad (2.6)$$

where ΔF and $\Delta \delta$ represent the variations in force and displacement measured within the elastic region of the experimental curve. In the present work, these values were determined considering the interval between 10% and 50% of the maximum force reached during the test, in order to reduce the influence of nonlinear effects and possible contact phenomena.

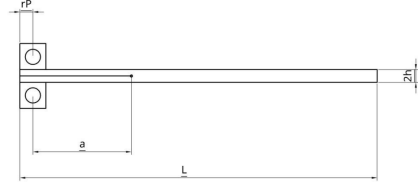
The determination of the flexural elastic modulus is fundamental in the context of this study, as this parameter is subsequently used in the calculation of the interlaminar fracture toughness for the different propagation modes analyzed and in the calibration of the numerical models developed in the following chapters.

2.5.2 Double Cantilever Beam (DCB) – Test Theory

Double Cantilever Beam (DCB) specimens represent one of the most widely used experimental configurations for the determination of interlaminar fracture toughness in Mode I in laminated composite materials [10].



(a) Real testing configuration.



(b) Geometrical scheme of the Double Cantilever Beam specimen

Figure 2.3: Experimental setup of the Double Cantilever Beam (DCB) test.

The DCB specimen consists of a pre-cracked laminate characterized by an initial crack length a_0 , extending along the interlaminar interface. Loading blocks are bonded at the specimen ends, through which a tensile force perpendicular to the delamination plane is applied, as illustrated in Figure 2.3.

During the test, the vertical displacement at the loading points and, in some experimental configurations, the rotation angle of the specimen arms measured by inclinometers are monitored. These quantities allow the determination of the interlaminar fracture toughness in Mode I, G_{Ic} , through different energetic approaches described below.

Assuming linear elastic behavior and modeling each half of the specimen as a clamped beam subjected to bending, the displacement δ at the loading point can be expressed as [11]:

$$\delta = \frac{2Fa^3}{3E_f I} \quad (2.7)$$

where F is the load applied at each loading block, a is the crack length, E_f is the flexural Young's modulus of the laminate, and I is the second moment of area of the cross-section.

Introducing the concept of compliance C , defined as the ratio between displacement and applied load:

$$C = \frac{\delta}{F} \quad (2.8)$$

one obtains:

$$C = \frac{2a^3}{3E_f I} = \beta a^3 \quad \text{with} \quad \beta = \frac{2}{3E_f I} \quad (2.9)$$

The derivative of the compliance with respect to the crack length is:

$$\frac{dC}{da} = 3\beta a^2 \quad (2.10)$$

Recalling the energetic formulation of Linear Elastic Fracture Mechanics, the interlaminar fracture toughness in Mode I can be calculated as:

$$G_{Ic} = \frac{F^2}{2B} \frac{dC}{da} \quad (2.11)$$

where B represents the specimen width. Substituting the expression of the second moment of area $I = \frac{1}{12} B h^3$, an explicit formulation of G_{Ic} under ideal assumptions is obtained:

$$G_{Ic} = \frac{12F^2 a^2}{B^2 h^3 E_f} \quad (2.12)$$

This expression provides a theoretical estimate of the interlaminar fracture toughness in Mode I under the assumption of linear elastic behavior and cubic dependence of compliance on crack length.

In addition to the compliance-based method, fracture toughness can also be evaluated using inclinometer measurements applied to the specimen arms. In this case, G_{Ic} is expressed as [12]:

$$G_{Ic} = \frac{2F\theta}{B} \quad (2.13)$$

where θ represents the average opening angle of the arms, calculated as:

$$\theta = \frac{|\theta_1| + |\theta_2|}{2} \quad (2.14)$$

A further approach for determining G_{Ic} is the energetic area-based method, which consists of evaluating the variation of elastic energy absorbed by the system between two successive crack propagation states [13]:

$$G_{Ic} = \frac{\Delta U}{\Delta A} = \frac{\Delta U}{B \Delta a} \quad (2.15)$$

where the energy variation ΔU is estimated from the measured force and displacement values at crack propagation events according to:

$$\Delta U = \frac{F_1 u_2 - F_2 u_1}{2} \quad (2.16)$$

as schematically illustrated in Figure 2.4.

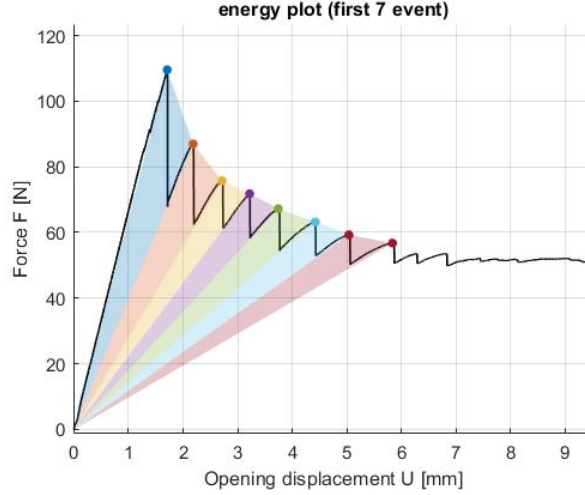
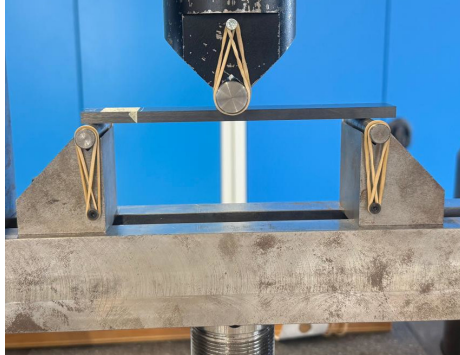
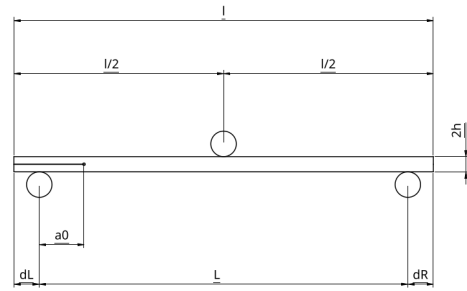


Figure 2.4: Force–displacement curve; triangular areas represent the energy variation ΔU used to compute G_{Ic} .

2.5.3 End-Notched Flexure (ENF) – Test Theory



(a) Real ENF testing configuration



(b) Geometrical scheme of the ENF specimen

Figure 2.5: End-Notched Flexure (ENF) test configuration: (a) real experimental setup; (b) geometrical scheme.

The End-Notched Flexure (ENF) test represents one of the most widely used experimental configurations for the determination of interlaminar fracture toughness in Mode II, G_{IIc} , in fiber-reinforced laminated composite materials [14].

The ENF specimen consists of a pre-cracked laminate characterized by an initial crack length a_0 , supported on two lower supports and loaded from above through a central pin, as shown in Figure 2.5. This configuration induces a stress state predominantly governed by in-plane shear within the laminate, promoting relative sliding between the delaminated surfaces and allowing the characterization of the material resistance to damage propagation in Mode II.

Assuming linear elastic behavior and modeling the specimen as a beam subjected to three-point bending, beam theory allows the vertical displacement δ at the loading point to be expressed as [14]:

$$\delta = \frac{F(3a_0^3 + 2L^3)}{8Bh^3E_f} \quad (2.17)$$

where:

- F is the applied force,
- a_0 is the initial crack length,
- L is the span between the lower supports,
- B is the specimen width,
- h is the laminate thickness,
- E_f is the flexural Young's modulus.

Introducing the compliance C , defined as:

$$C = \frac{\delta}{F} = \frac{3a_0^3 + 2L^3}{8Bh^3E_f} \quad (2.18)$$

the interlaminar fracture toughness in Mode II can be derived using the energetic formulation of Linear Elastic Fracture Mechanics as:

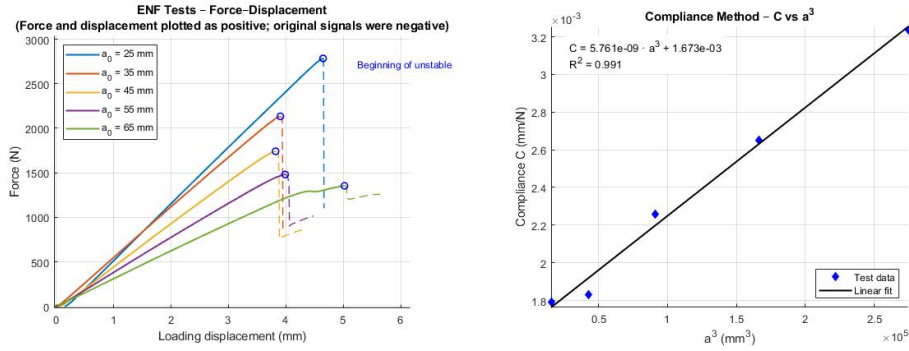
$$G_{IIc} = \frac{F^2}{2B} \frac{dC}{da} = \frac{9a_0^2 F^2}{16B^2 h^3 E_f} \quad (2.19)$$

where F represents the critical load corresponding to the onset of delamination.

However, unlike the DCB test, the ENF test is characterized by intrinsically unstable crack propagation. This instability manifests experimentally through sudden drops in the force–displacement curve, associated with rapid crack advances.

For this reason, the experimental determination of G_{IIc} is generally not performed by following the crack evolution on a single specimen. Instead, it is obtained by performing multiple tests on specimens characterized by different initial crack lengths a_0 .

The combined analysis of the force–displacement curves obtained for different values of a_0 allows the identification of the onset of unstable crack propagation and the derivation of a unique relationship between the system compliance and the crack length. This behavior is represented by comparing the force–displacement curves as a function of the initial crack length, as shown in Figure 2.6a, highlighting the different stiffness levels and the occurrence of unstable propagation. From the measured critical values, it is then possible to construct the calibration law required by the compliance method, representing C as a function of a^3 , as shown in Figure 2.6b.



(a) Force–displacement curves for different initial crack lengths a_0 .

(b) Compliance method: representation of C as a function of a^3 .

Figure 2.6: Experimental response of the ENF test and construction of the calibration relationship for the calculation of G_{IIc} using the compliance method: (a) force–displacement curves as a function of a_0 ; (b) calibration law $C(a^3)$.

According to ASTM D7905 [15], the dependence of compliance on crack length can be approximated through a polynomial law of the form:

$$C(a) = ma^3 + c_0 \quad (2.20)$$

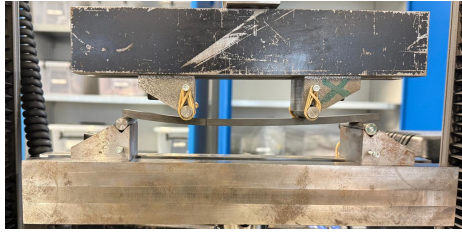
where the coefficients m and c_0 are determined through linear regression of the experimental data obtained from the tested specimens. This relationship forms the basis for the calculation of G_{IIc} using the compliance method adopted in this work.

2.5.4 Four-Point End-Notched Flexure (4ENF)

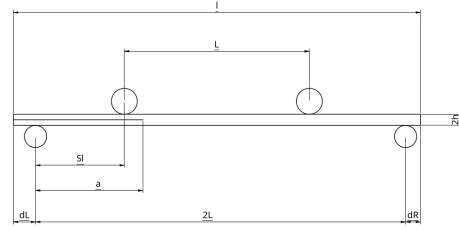
In the traditional End-Notched Flexure (ENF) test, as discussed in the previous section, delamination propagation typically occurs in an intrinsically unstable manner. This behavior makes it difficult to determine the interlaminar fracture toughness in Mode II throughout the entire crack evolution, since crack growth cannot be followed continuously on a single specimen.

To overcome this limitation, the *Four-Point End-Notched Flexure* (4ENF) test was proposed, allowing a substantially stable delamination propagation. The main advantage of the 4ENF test lies in the possibility of determining the interlaminar fracture toughness G_{IIc} at multiple points during a single test, unlike the ENF test, which requires several specimens with different initial crack lengths.

However, the 4ENF test is less widely used in experimental practice compared to ENF, as the values of G_{IIc} obtained may significantly differ from those measured using the ENF configuration. These differences are attributed to several factors, including interlaminar friction effects, which play a particularly relevant role in this testing configuration [16].



(a) Real experimental configuration of the 4ENF test.



(b) Geometrical scheme of the 4ENF specimen.

Figure 2.7: Four-Point End-Notched Flexure (4ENF) test: (a) experimental setup; (b) geometrical configuration and dimensional parameters.

From a geometrical standpoint, the 4ENF test differs from the ENF test due to the presence of two loading points instead of one. The specimen is therefore subjected to four-point bending, as illustrated in Figure 4.33, showing the testing configuration and the main geometrical dimensions.

This loading configuration allows a mechanical response characterized by an approximately constant load plateau during delamination propagation. This behavior is clearly highlighted by the presence of a plateau in the force–displacement curve of the 4ENF test, indicative of stable crack growth, as shown in Figure 2.8.

Assuming initially the absence of interlaminar friction, the interlaminar fracture toughness in Mode II can be derived from Griffith’s energetic theory. In this framework, the energy release rate is defined as:

$$G_{IIc} = -\frac{1}{B} \frac{\partial \Pi}{\partial a} \quad (2.21)$$

where B is the specimen width, Π the total potential energy of the system, and a the delamination length. For the 4ENF test, this formulation leads to the expression [16]:

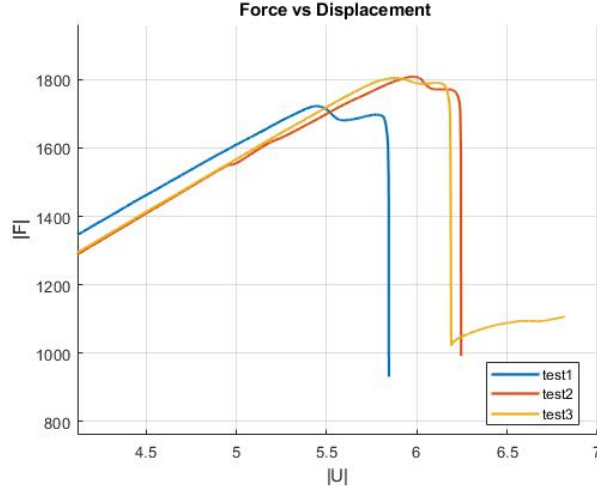


Figure 2.8: Force–displacement curve of the 4ENF test.

$$G_{IIc} = \frac{9S_l^2 F^2 \eta}{16h^3 B^2 E_f} \quad (2.22)$$

where S_l is the distance between the support and the loading point on the delaminated side, F the applied load, h the specimen thickness, E_f the flexural Young's modulus of the laminate, and η is a correction coefficient accounting for the stress state, taking the value $\eta = 1$ under plane stress conditions and $\eta = 1 - \nu_{xz}\nu_{zx}$ under plane strain conditions.

In the 4ENF test, the effect of interlaminar friction cannot be neglected, as it significantly contributes to energy dissipation during delamination propagation. Considering this contribution, the energy release rate can be expressed as:

$$G_{IIc} = -\frac{1}{B} \left(\frac{\partial W_{\text{ext}}}{\partial a} - \frac{\partial W_{\text{el}}}{\partial a} - D_a \right) \quad (2.23)$$

where W_{ext} is the external work, W_{el} the elastic strain energy stored in the system, and D_a the dissipation term associated with interlaminar friction. Introducing the friction coefficient α , the corrected expression for the Mode II fracture toughness becomes [16]:

$$G_{IIc} = \frac{(3S_l - 4h\alpha)^2}{16h^3 B^2 E_f} \eta F^2 \quad (2.24)$$

The above formulation highlights how interlaminar friction plays a key role in the determination of the fracture toughness measured by the 4ENF test. This aspect represents one of the main differences compared to the ENF test and justifies the discrepancies observed between the G_{IIc} values obtained from the two experimental configurations.

In the present work, the 4ENF test is used to analyze interlaminar Mode II fracture behavior under stable propagation conditions, allowing a more detailed comparison with the numerical results presented in the following chapters.

2.6 Numerical modeling of delamination

In order to validate and interpret the experimental results of the tests presented in the following chapters, it was necessary to implement numerical models capable of simulating delamination propagation in laminated composite materials. Numerical modeling indeed makes it possible to analyze, in a controlled manner, the mechanisms of crack initiation and growth, providing a useful tool for comparison with experimental data and for supporting the interpretation of the results.

Within the context of finite element analysis of delamination, several numerical approaches have been proposed in the literature. Among these, three methodologies are the most commonly adopted: the *Virtual Crack Closure Technique* (VCCT), the *Cohesive Zone Model* (CZM), and the *Extended Finite Element Method* (XFEM). A comparative overview of these approaches applied to DCB specimens is reported in [17].

In the present work, the focus is exclusively on the VCCT and CZM methods, since these were effectively adopted for the numerical simulations. The following sections therefore provide a theoretical summary of these approaches, sufficient to understand the modeling strategies and the numerical results presented later.

2.6.1 Virtual Crack Closure Technique (VCCT)

The Virtual Crack Closure Technique is a method based on the energetic principle originally introduced by Irwin, according to which the energy required to propagate a crack is equivalent to that required to virtually close it. In VCCT, the crack is already present in the numerical model and its propagation occurs along a known and predefined path.

The method directly uses the nodal forces and nodal displacements provided by the finite element analysis to estimate the energy release rate associated with crack advance.

Referring to the two-dimensional scheme shown in Figure 2.9, the contributions of Mode I and Mode II can be expressed, respectively, as [17]:

$$G_I = \frac{1}{2\Delta a} F_{y,i} \Delta v_{k,j} \quad (2.25)$$

$$G_{II} = \frac{1}{2\Delta a} F_{x,i} \Delta u_{k,j} \quad (2.26)$$

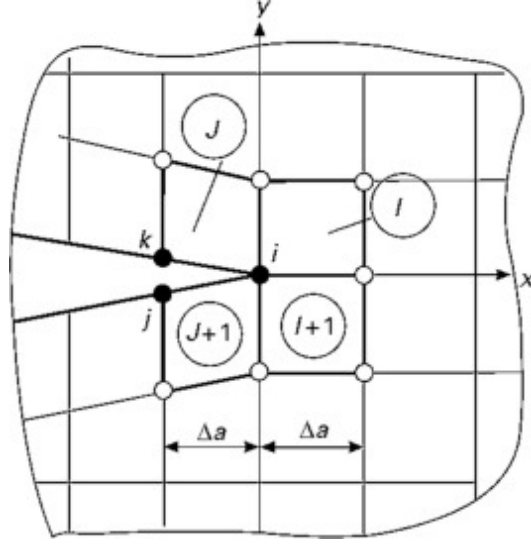


Figure 2.9: Two-dimensional scheme of the VCCT method for four-node quadrilateral elements. Image reproduced from [17].

where $F_{x,i}$ and $F_{y,i}$ are the components of the nodal forces applied at the node immediately ahead of the crack tip, while $\Delta u_{k,j}$ and $\Delta v_{k,j}$ represent the relative displacements of the nodes located immediately behind the propagation front. The term Δa indicates the virtual increment of crack length, which is related to the finite element size.

The VCCT method is effective and relatively simple to implement, but it presents some intrinsic limitations. In particular, it requires an initial pre-crack, a mesh that is accurately refined near the delamination region, and coincident nodes on the upper and lower surfaces of the interface. Moreover, the crack propagation path must be known *a priori*.

2.6.2 Cohesive Zone Model (CZM)

A conceptually different approach is represented by the Cohesive Zone Model, in which the crack is modeled as a region of progressive damage rather than as a sharp geometric discontinuity. In CZM, delamination is described by cohesive elements characterized by a traction–separation constitutive law.

The initial elastic behavior of the cohesive zone can be expressed as:

$$\mathbf{t} = \mathbf{K} \boldsymbol{\delta} \quad (2.27)$$

where $\mathbf{t}$ is the traction vector, $\boldsymbol{\delta}$ the separation vector, and $\mathbf{K}$ the stiffness matrix. Assuming no coupling between fracture modes, the stiffness matrix takes a diagonal form [17]:

$$\mathbf{K} = \begin{bmatrix} K_n & 0 & 0 \\ 0 & K_s & 0 \\ 0 & 0 & K_t \end{bmatrix} \quad (2.28)$$

Damage initiation is often described through a maximum stress criterion:

$$\max \left(\frac{t_n}{t_n^0}, \frac{t_s}{t_s^0}, \frac{t_t}{t_t^0} \right) = 1 \quad (2.29)$$

Once damage is initiated, the cohesive response is degraded through a damage variable D , ranging from 0 (undamaged material) to 1 (complete separation of the surfaces):

$$\mathbf{t} = (1 - D) \mathbf{t}_0 \quad (2.30)$$

Damage evolution is governed by the critical fracture energy G_c , and complete separation occurs when the dissipated energy equals this value.

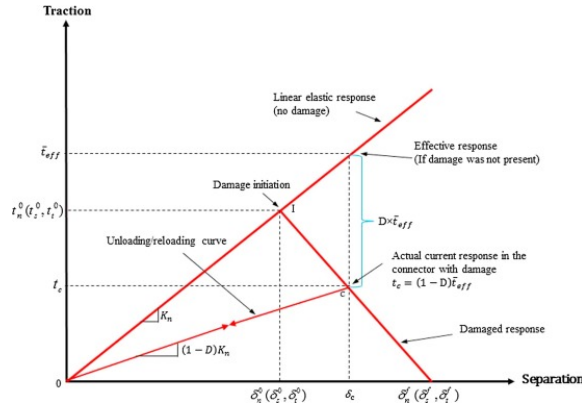


Figure 2.10: Bilinear traction–separation law of the cohesive model (CZM). Image reproduced from [17].

Figure 2.10 shows an example of a bilinear traction–separation law, highlighting the stages of elastic response, damage, and complete separation.

The main advantage of CZM lies in the ability to simulate both initiation and propagation of delamination without the need to introduce an initial pre-crack. However, the method is computationally more demanding and requires a careful definition of a larger number of material parameters.

Chapter 3

Experimental Tests

In this chapter, the results obtained from the experimental campaign conducted on the composite material are presented and discussed. The mechanical behaviour of the laminate is investigated through different testing configurations, including tensile and interlaminar fracture tests (DCB, ENF and 4ENF). The objective is to characterize both the in-plane elastic response and the interlaminar fracture toughness of the material under Mode I and Mode II loading conditions.

3.1 Material and Experimental Equipment

3.1.1 Material

The composite material used for the experimental campaign is a unidirectional carbon fiber reinforced composite with a thermosetting epoxy matrix.

The material was provided in the form of cured laminate panels manufactured from prepreg plies. The laminates were produced following industrial manufacturing procedures representative of aeronautical composite structures.

Airbus Atlantic provided the following cured laminate panels:

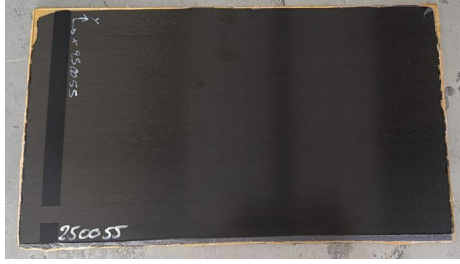
- $[0]_{32}$
- $[0]_{32}$ with a release film insert
- $[60/-60]_{2s}$

The $[0]_{32}$ laminates were used for interlaminar fracture tests (DCB, ENF and 4ENF). The non-adhesive insert allowed the introduction of a predefined initial delamination length.

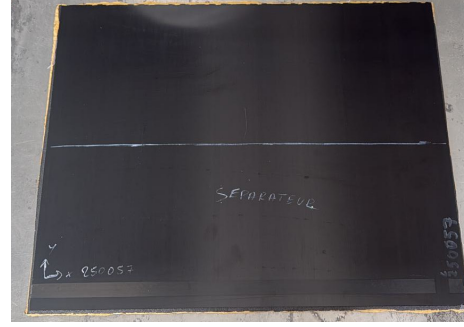
The $[60/-60]_{2s}$ laminate was used for tensile characterization.

In addition to the cured panels, a roll of prepreg material was also supplied, from which additional specimens were manufactured when required.

Representative images of the supplied laminate panels and the prepreg roll are shown in Figure 3.1.



(a) $[0]_{32}$ laminate panel.



(b) $[0]_{32}$ laminate with insert.



(c) $[60/-60]_{2s}$ laminate panel.



(d) Prepreg material roll used for specimen manufacturing.

Figure 3.1: Composite material supplied by Airbus Atlantic: cured laminate panels and prepreg roll used for specimen manufacturing.

3.1.2 Testing Machine

All experimental tests were performed using an INSTRON 5969 universal testing machine with a maximum load capacity of 50 kN.

The machine is equipped with a displacement-controlled loading system and an integrated load cell for force measurement. This configuration allows

precise control of the applied displacement rate and accurate acquisition of force–displacement curves during the different mechanical tests.

A representative image of the testing machine is shown in Figure 3.2.

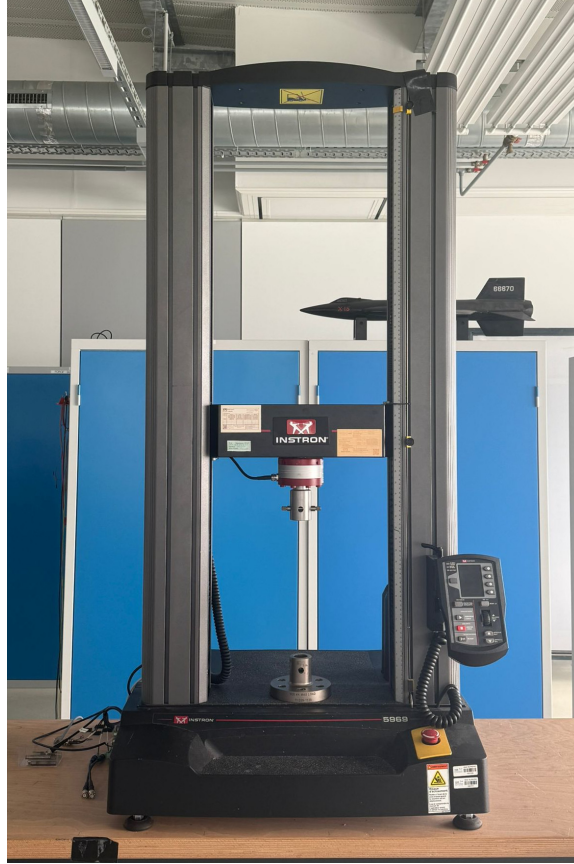


Figure 3.2: INSTRON 5969 testing machine.

3.2 Three-Point Bending (3PB) — Flexural Modulus Identification

3.2.1 Specimen and Experimental Setup

A preliminary three-point bending test was performed with the aim of determining the flexural elastic modulus of the material. To this end, three rectangular specimens were extracted from a laminate panel with fibers oriented at 0° . The geometrical dimensions of the specimens are reported in Figure 3.3 and in Table 3.1. Each specimen consisted of 32 plies.

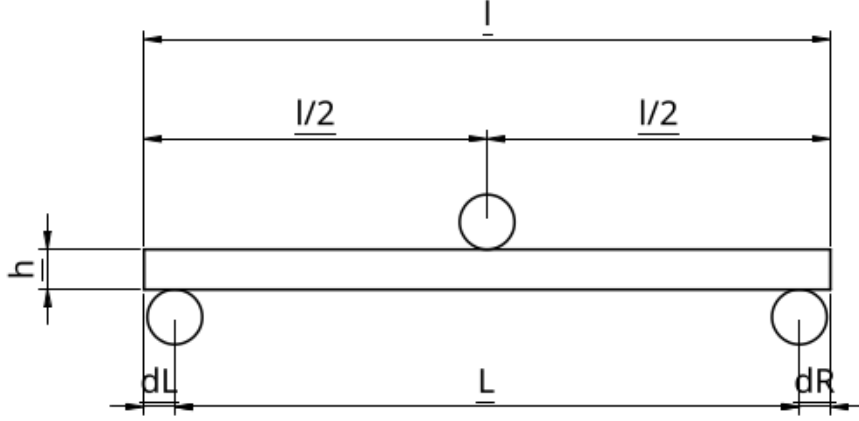


Figure 3.3: Geometrical dimensions of the three-point bending specimen.

Table 3.1: 3PB dimensions

Sample	L [mm]	B [mm]	h [mm]	l [mm]	d_L [mm]	d_R [mm]
3PB	91	10	5.85	100	4.5	4.5

A maximum displacement of 10 mm was imposed on the specimens, applied at a crosshead speed of 2 mm/min. In order to determine the flexural modulus, the vertical displacement of the specimen and the applied load were measured using the internal sensors of the testing machine.

3.2.2 Data Reduction and Flexural Modulus Evaluation

The flexural modulus was calculated using Equation (2.6). The force and displacement points used for the evaluation are shown in Figure 3.4, corresponding to the interval between 10% and 50% of the peak load F_{peak} .

In addition, the compressive failure stress of the fibers in the 0° direction was evaluated using the following relation:

$$\sigma_{f,11} = \frac{F_{\text{peak}} L h}{8 I_z} \quad \text{with} \quad I_z = \frac{1}{12} b h^3 \quad (3.1)$$

where F_{peak} represents the maximum load reached by the specimen prior to failure.

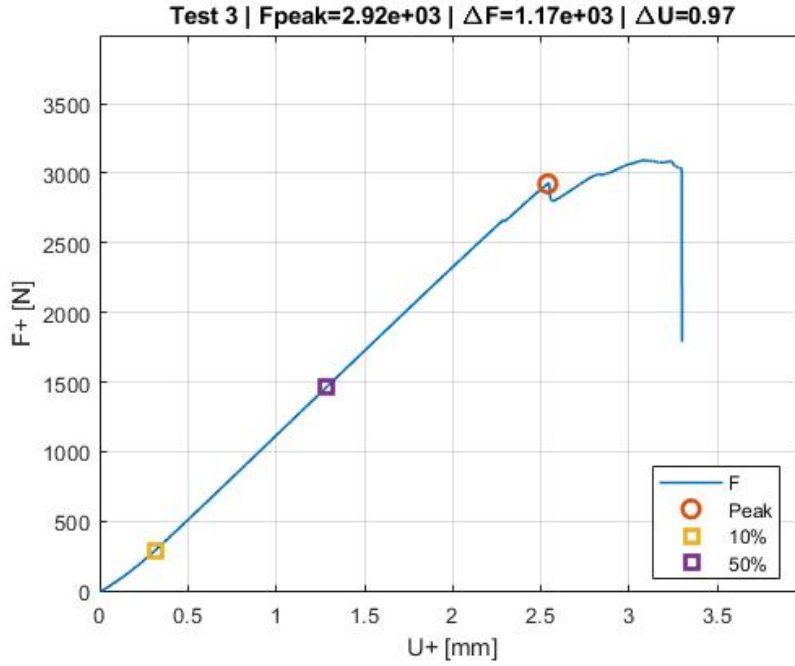


Figure 3.4: Force–displacement curve highlighting the 10%–50% F_{peak} interval used for the calculation of the flexural modulus.

3.2.3 Results and Discussion

The force–displacement curves obtained from the three-point bending tests are reported in Figure 3.5. All three specimens exhibit a predominantly linear elastic response in the initial loading phase, followed by a sudden load drop associated with compressive failure on the upper surface of the beam. The similarity of the initial slopes indicates a consistent bending stiffness among the tested specimens and confirms the repeatability of the experimental procedure.

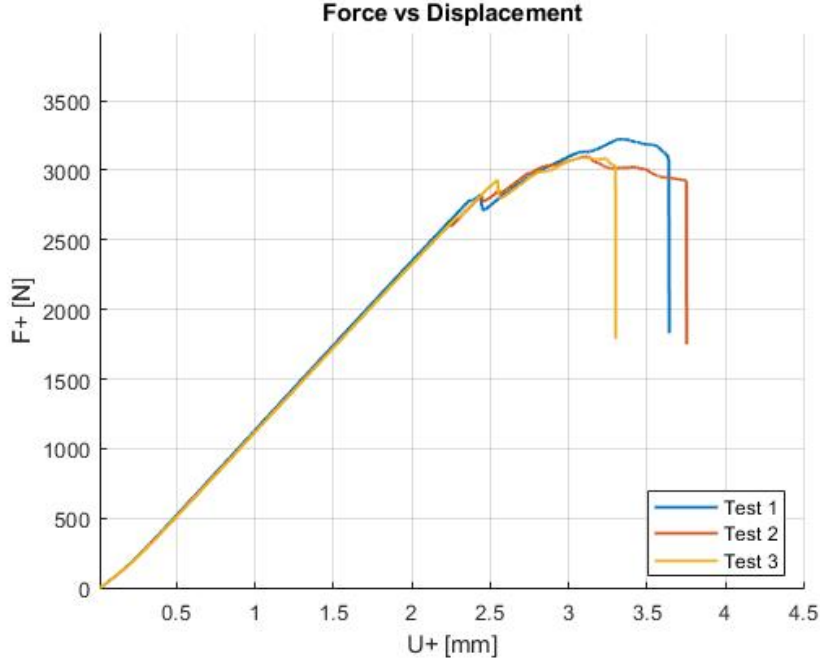


Figure 3.5: Force–displacement curves obtained from the three-point bending tests.

The experimental values of the flexural modulus E_f , peak load F_{peak} , and compressive failure stress $\sigma_{f,11}$ are summarised in Table 3.2. The identified flexural modulus shows limited dispersion between specimens, with values ranging from 113283 MPa to 114288 MPa and an average value of 113697 MPa. The relative variation remains below 1%, demonstrating a robust identification of the elastic stiffness in bending.

Table 3.2: Experimental results of the three-point bending tests.

Specimen	E_f [MPa]	F_{peak} [N]	$\sigma_{f,11}$ [MPa]
1	114288	2819	-1124
2	113283	2812	-1216
3	113522	2925	-1166
average	113697	–	-1138

The compressive failure stress $\sigma_{f,11}$, calculated from the peak load, shows a moderate variability between specimens. This dispersion is attributed to local damage mechanisms and fibre–matrix interactions governing compressive failure, which are typically more sensitive to microstructural imperfections than the elastic response.

The average flexural modulus obtained from the three-point bending

test is in good agreement with the nominal longitudinal modulus provided in the material datasheet for a $[0]_{32}$ laminate. Since all fracture specimens (DCB, ENF and 4ENF) were extracted from the same laminate panel, the identified modulus is considered representative of the longitudinal bending stiffness entering the analytical formulations used for the evaluation of G_{Ic} and G_{IIc} .

Therefore, the three-point bending test provides a consistent and experimentally validated elastic parameter that will be employed as input in the subsequent interlaminar fracture toughness calculations.

3.3 Tensile Test ($\pm 60^\circ$) — In-Plane Elastic Properties

3.3.1 Specimen and Experimental Setup

The tensile test was performed on an 8-ply symmetric laminate characterized by an alternating $\pm 60^\circ$ stacking sequence. The objective of this test is to analyse the mechanical response of the laminate when subjected to loading conditions not aligned with the principal fibre directions, and to determine its apparent elastic properties in the global reference frame.

In this work, the global x -axis is defined as the longitudinal direction of the specimen and corresponds to the loading direction, while the global y -axis represents the transverse direction.

The specimen geometry is shown in Figure 3.6 and its main dimensions are reported in Table 3.3.



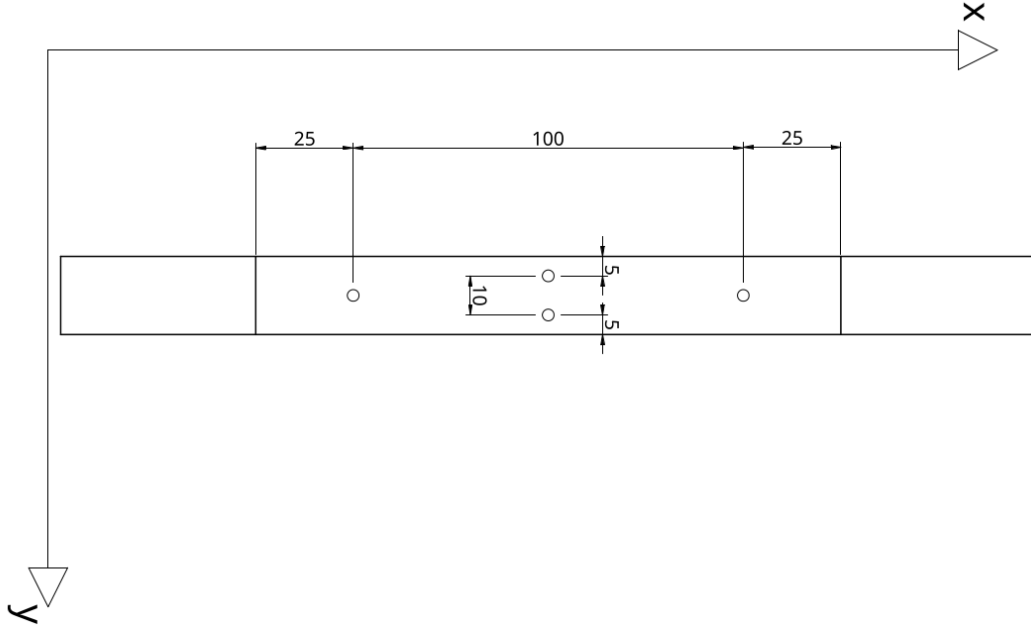
Figure 3.6: Geometry of the $\pm 60^\circ$ tensile specimen.

Table 3.3: Geometrical parameters of the specimen.

L	B	h	l	x_L	x_R
250	20	1.45	150	50	50

The test consists of applying a uniaxial tensile load along the global x -direction of the specimen.

Axial and transverse strains were measured using an optical tracking system. Four reference dots were manually placed on the specimen surface: two aligned along the longitudinal direction (x -axis) and two aligned along the transverse direction (y -axis). The initial configuration of the dots is illustrated in Figure 3.7, which shows both the CAD representation and the real specimen used during the experiments.



(a) CAD representation of the reference dots configuration.



(b) Real specimen with the optical tracking reference dots.

Figure 3.7: Reference dots used for the optical measurement of axial and transverse strains.

The evolution of the dot positions during the test was recorded through an optical acquisition system, illustrated in Figure 3.8. By tracking the relative displacement of each pair of dots, it was possible to compute the axial strain ε_x and the transverse strain ε_y according to

$$\varepsilon = \frac{L - L_0}{L_0} \quad (3.2)$$

where L_0 is the initial distance between the dots and L is the instantaneous distance during loading.



Figure 3.8: Optical acquisition system used to track the displacement of the reference dots during the tensile test.

3.3.2 Data Reduction and Identification of Elastic Parameters

The nominal axial stress was computed as

$$\sigma = \frac{F}{A} \quad (3.3)$$

where F is the applied axial force and $A = b \cdot h$ is the cross-sectional area of the specimen.

The apparent Young's modulus of the laminate in the loading direction was determined from the linear elastic portion of the stress-strain curve as

$$E_x = \frac{d\sigma}{d\varepsilon_x} \quad (3.4)$$

while the in-plane Poisson's ratio was evaluated as

$$\nu_{xy} = -\frac{\varepsilon_y}{\varepsilon_x} \quad (3.5)$$

where ε_x is the axial strain and ε_y is the transverse strain.

3.3.3 Results and Discussion

The force–displacement behaviour of the tested specimens is presented in Figure 3.9.

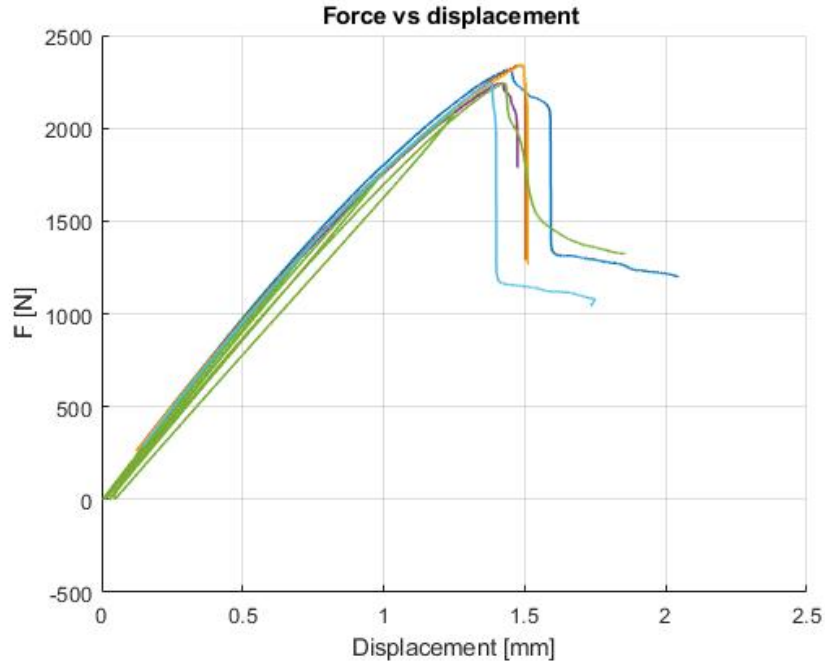


Figure 3.9: Force–displacement curves obtained from the tensile tests.

The stress–strain response in both longitudinal and transverse directions is shown in Figure 3.10, where the initial linear region used for modulus evaluation can be observed.

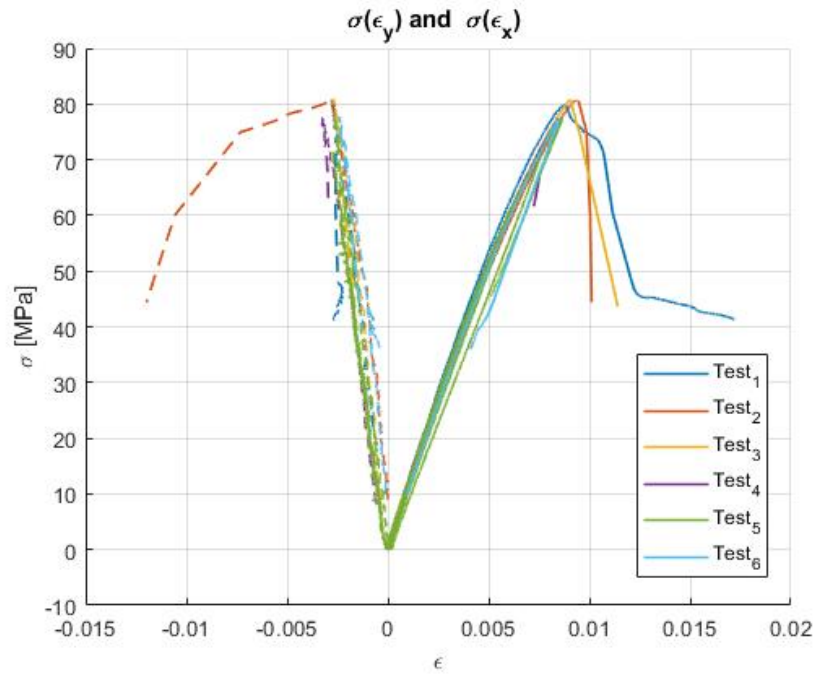
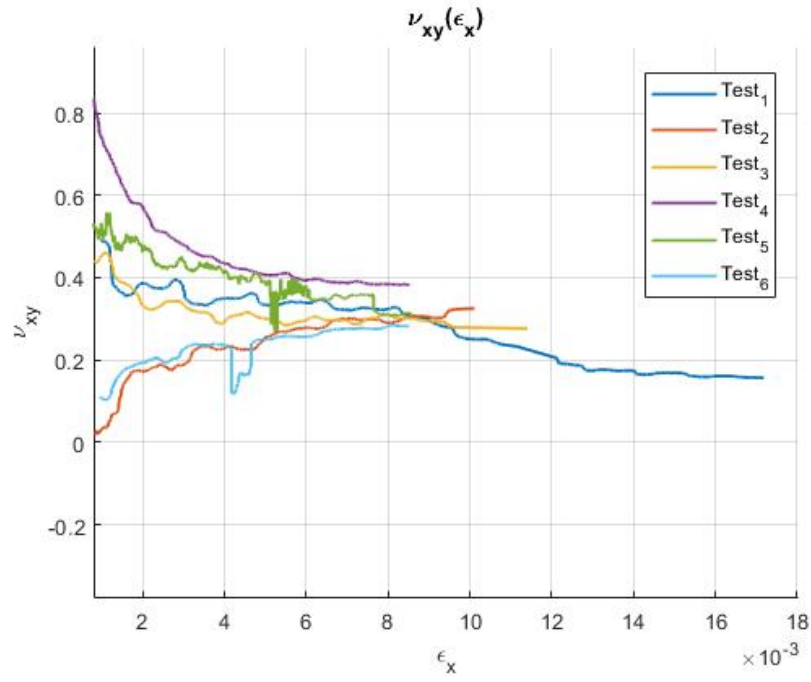
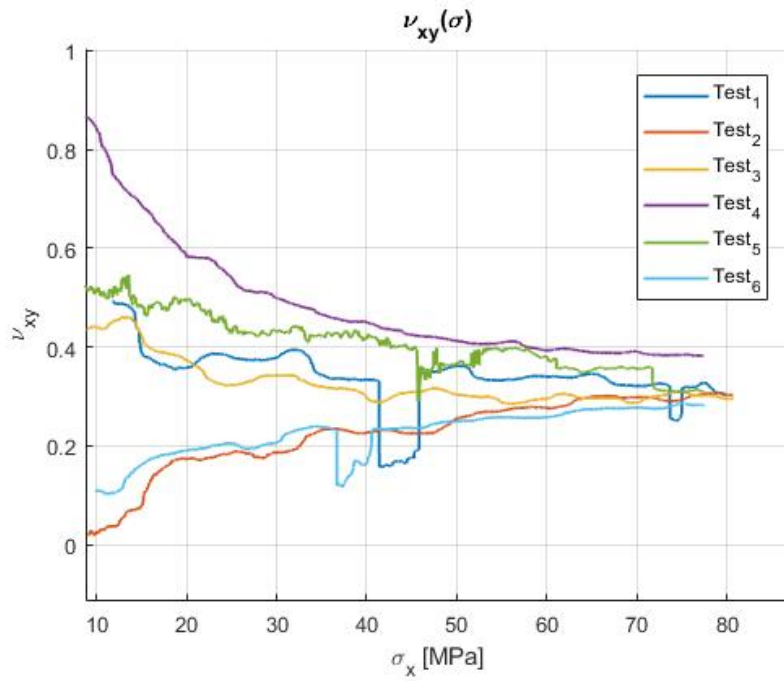


Figure 3.10: Stress–strain behaviour in the longitudinal (x) and transverse (y) directions.

The evolution of Poisson’s ratio as a function of stress and strain is reported in Figure 3.11.



(a) ν_{xy} as a function of axial strain.



(b) ν_{xy} as a function of σ_x .

Figure 3.11: Evolution of Poisson's ratio ν_{xy} during the tensile tests as a function of strain and stress.

The obtained values of E_x and ν_{xy} are summarised in Table 3.4.

Table 3.4: Summary of the Poisson’s ratio ν_{xy} and longitudinal Young’s modulus E_x obtained from the tensile $\pm 60^\circ$ tests.

Test	ν_{xy}	E_x [MPa]
Test 1	0.31911	10655
Test 2	0.28633	9899
Test 3	0.26763	10009
Test 4	0.31995	10155
Test 5	0.35813	10576
Test 6	0.29076	10206
Average	0.3070	10250

From the force–displacement curves shown in Figure 3.9, it can be observed that the six tests are overall consistent in the elastic region. The initial slopes of the loading branches are very similar, which indicates a good repeatability of the experimental procedure and a reliable identification of the elastic response.

Some differences appear close to the maximum load and in the post-peak region. This behaviour is expected, since damage initiation and propagation are influenced by local imperfections and fibre–matrix interactions. For this reason, the dispersion observed after the peak does not significantly affect the evaluation of the elastic properties.

Regarding the Poisson’s ratio ν_{xy} (Figure 3.11), the curves show an unstable behaviour at very low strain levels. This is mainly due to the fact that the ratio $-\varepsilon_y/\varepsilon_x$ becomes sensitive when ε_y is close to zero. After this initial region, the values progressively stabilize and remain within a relatively narrow range. The average value is approximately 0.31, which is coherent with the expected transverse contraction behaviour of a $\pm 60^\circ$ laminate. Although a certain dispersion between specimens is visible, the overall trend is consistent across all tests.

The Young’s modulus in the axial direction, identified from the linear regression in the 10–50% peak force interval, results in an average value of about 10 250 MPa (see Table 3.4). The similarity of the elastic slopes among the different tests confirms that this value is representative of the material behaviour.

To further support the experimental identification, the equivalent elastic properties of the laminate were also estimated using a MATLAB routine based on Classical Laminate Theory, provided during the internship. The routine takes as input the ply material properties, stacking sequence and ply thickness in order to compute the homogenized elastic properties of the

laminate. For the tested $\pm 60^\circ$ configuration, it predicts an axial modulus $E_x = 1.039 \times 10^4$ MPa and a Poisson's ratio $\nu_{xy} = 0.3114$. These values are in very good agreement with the experimental averages, namely $E_x = 10250$ MPa and $\nu_{xy} = 0.3070$, with a difference below approximately 2%.

This modulus is significantly lower than the one expected for a unidirectional composite with fibres aligned at 0° . This difference is fully coherent with the $\pm 60^\circ$ stacking sequence of the tested laminate. Since the fibres are inclined with respect to the loading direction, the applied axial load is not carried purely along the fibre direction. Instead, part of the load is transferred through shear and matrix deformation, which reduces the effective longitudinal stiffness compared to a laminate reinforced only in the loading direction.

Overall, the experimental results are mechanically consistent with the laminate configuration and provide a reliable estimation of its elastic properties.

3.4 Double Cantilever Beam (DCB) — Mode I Interlaminar Fracture Toughness

3.4.1 Specimen and Experimental Setup

The objective of the present test is the determination of the critical Mode I interlaminar fracture toughness, G_{Ic} .

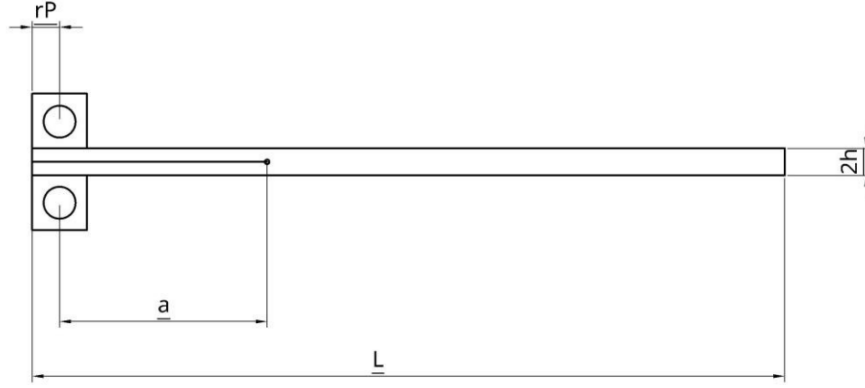


Figure 3.12: Geometrical scheme of the Double Cantilever Beam specimen

Table 3.5: DCB dimensions

Sample	a_0 [mm]	L [mm]	B [mm]	h [mm]	r_P [mm]
DCB	45.5	166	20	3	6

The specimen was extracted from a laminated panel with a fibre orientation of 0° , consisting of 32 plies. The geometric dimensions of the specimen are reported in Figure 3.12 and Table 3.5. The original panel contained a pre-cracked region obtained during manufacturing by inserting a Kapton film, in order to ensure a controlled initial crack along the interlaminar interface.

The loading blocks applied to the specimen for load introduction have a diameter of 12 mm.

In order to ensure experimental reliability and allow comparison between repeatable results, a total of five tests were performed on five geometrically identical specimens.

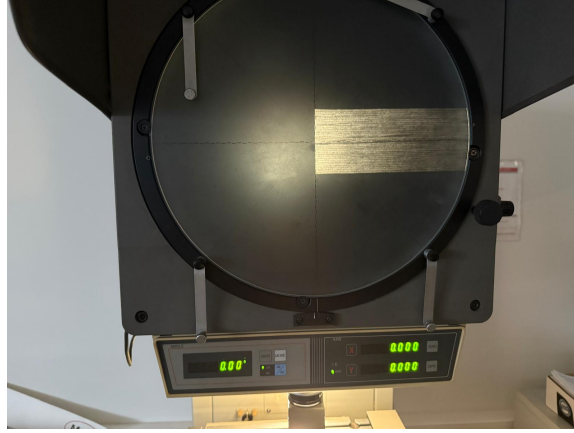


Figure 3.13: Microscope used to verify initial crack length

The initial crack length was verified during the pre-test phase using an optical microscope, as shown in Figure 3.13.

During the test, two types of sensors were employed: external inclinometers, used to measure the opening angle of the loading blocks, and displacement sensors integrated within the testing machine, used to measure the vertical opening displacement of the specimen.

The testing machine records the applied load. The operating conditions were set as follows:

- displacement rate: 2 mm/min
- maximum displacement: 50 mm
- maximum load: 20 kN

Given these operating parameters and the specimen dimensions, the machine provides as output the applied load (N), the opening displacement (mm), and the two opening angles θ (degrees) corresponding to the rotation of the specimen arms.

Figure 3.14 shows the force–displacement curves for the five tests, with load on the vertical axis and displacement on the horizontal axis.

3.4.2 Data Reduction and Fracture Toughness Evaluation

Four different approaches were adopted to determine the critical fracture toughness G_{Ic} , all derived from the theoretical framework described in Section 2.5.2.

- **Inclinometer-based method:** The value of G_{Ic} was calculated using the angle-based formulation reported in Eq. 2.13. The equation was applied to the final portion of the test. This region does not correspond

to a perfectly steady-state regime, as the evolution of G_I remains increasing; however, compared to the initial propagation stage, it is characterized by a more regular response and is largely free from the frequent energy jumps observed during unstable crack growth. The selected interval is highlighted in Figure 3.16.

- **Beam Theory method:** The value of G_{Ic} was determined according to the beam theory formulation described in Eq. 2.12.
- **Energy-based method:** G_{Ic} was calculated using the energy-based formulation reported in Eq. 2.15, where the elastic energy increment is computed according to Eq. 2.16. The crack length evolution was reconstructed by analysing the fracture surfaces of the specimens after testing, as shown in Figure 3.17.
- **Compliance method:** The value of G_{Ic} was calculated using the compliance-based formulation given in Eq. 2.11, with the derivative of the compliance obtained from Eq. 2.10, based on the same crack length evolution adopted for the energy-based approach.

3.4.3 Results and Discussion

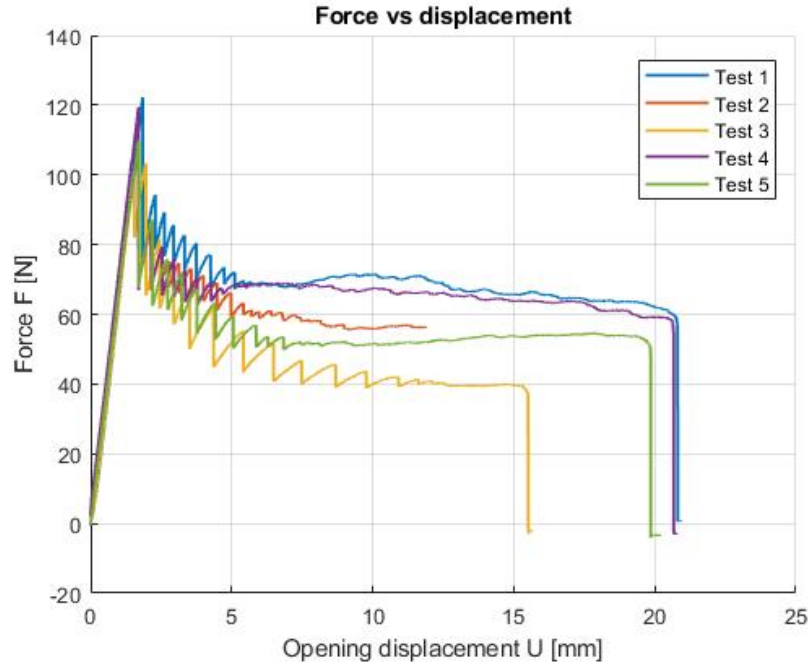


Figure 3.14: Force–displacement curves for the five DCB tests.

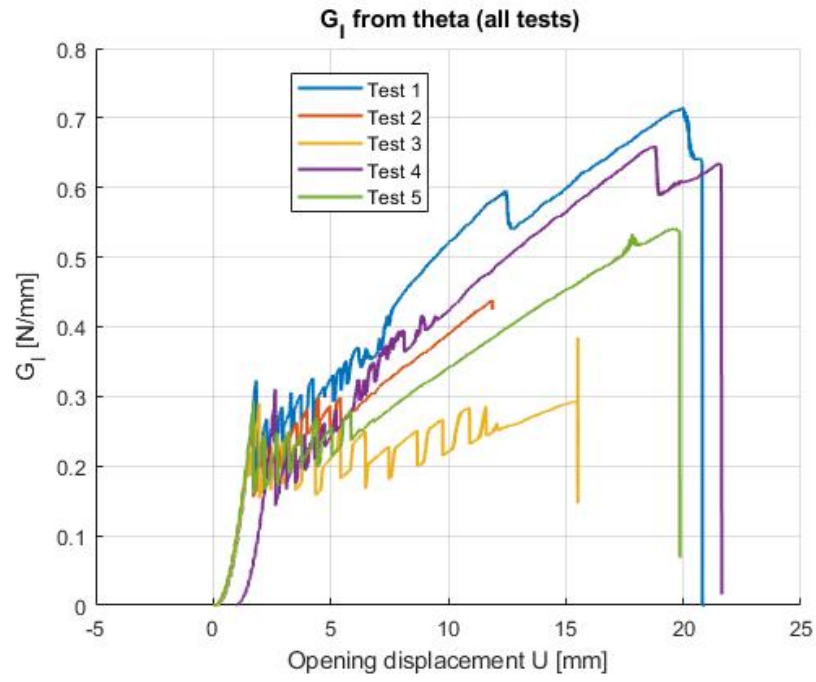


Figure 3.15: Evolution of G_{Ic} as a function of opening displacement.

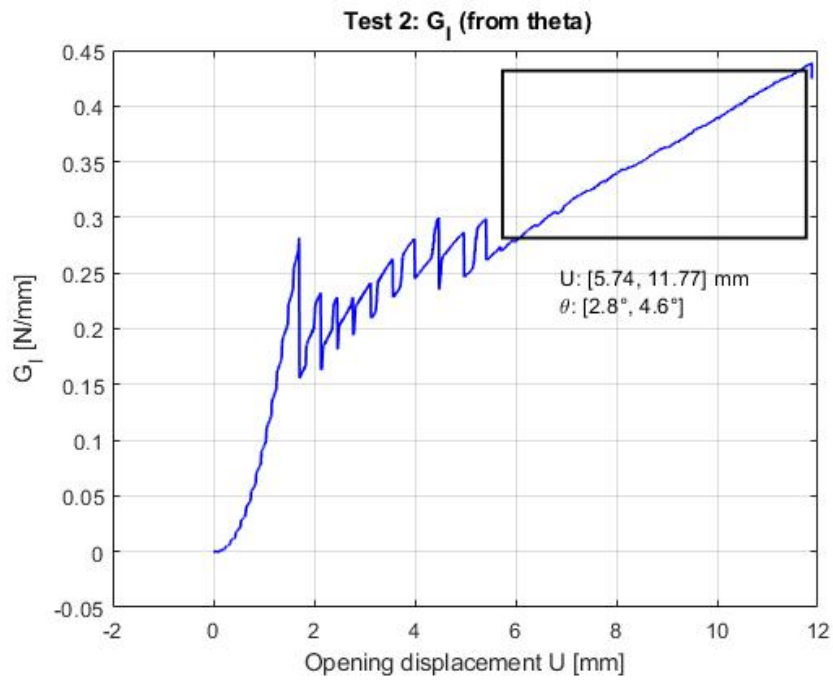


Figure 3.16: Evolution of G_{Ic} calculated using inclinometers.

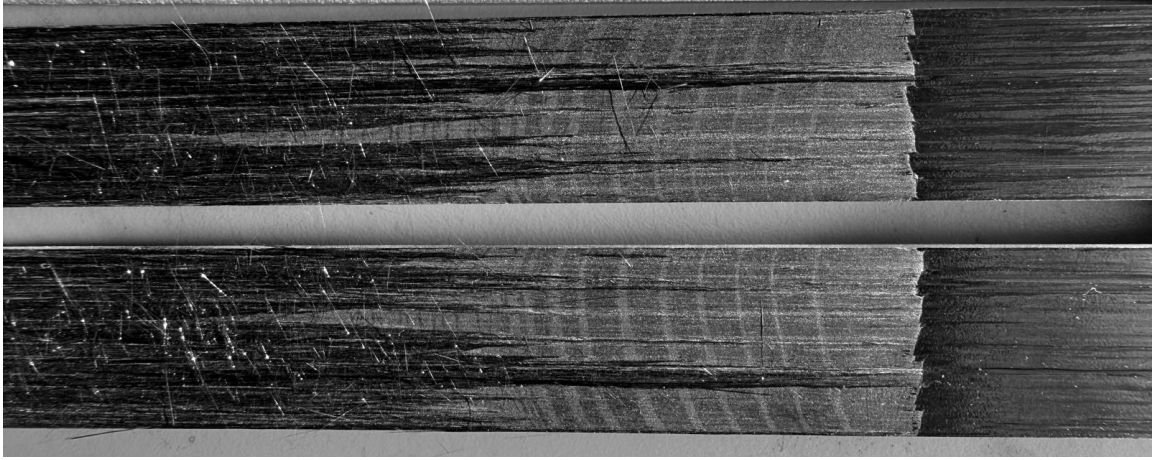


Figure 3.17: Fracture surfaces of the Test 1 of DCB specimens.

Table 3.6: G_{IC} Results

Test	$G_{IC}(\theta)$ Inclinometers [N/mm]	U interval used for $G_{IC}(\theta)$ [mm]	G_{IC} Beam theory [N/mm]	G_{IC} Area method [N/mm]	G_{IC} Compliance [N/mm]
Test 1	0.5776	7.55–19.30	0.3003	0.2701	0.2917
Test 2	0.3573	5.96–11.60	0.2425	(no data)	(no data)
Test 3	0.2734	12.33–15.28	0.2046	0.2400	0.2649
Test 4	0.5609	9.99–21.05	0.2863	0.2376	0.2595
Test 5	0.4067	6.85–19.16	0.2429	0.2402	0.2644
Average	0.4352		0.2553	0.2470	0.2704

From the analysis of the graphs (Figure 3.15), it can be observed that crack propagation is initially unstable, with the presence of energy jumps associated with sudden crack advances. In the later stages, before complete specimen opening, the crack growth evolves towards a more regular propagation regime.

Table 3.6 reports the average values obtained using the different calculation methods. It can be observed that, except for the inclinometer-based method, all approaches provide an average G_{IC} value of approximately 0.25 N/mm.

The difference observed for the inclinometer-based method is attributed to the different portion of the curve used for the calculation. For this method, only the final part of the test was considered, characterized by a more regular trend and less affected by energy jumps. This region exhibits higher G_{IC} values compared to the initial unstable propagation stage, as shown in Figure 3.16.

The selection of this portion of the curve was motivated by the need to

reduce the influence of sudden energy jumps associated with unstable crack advances. In the initial propagation stage, the presence of pronounced load drops introduces significant fluctuations in the measured rotation angles, leading to a less stable evaluation of G_{Ic} . By contrast, the final propagation region, where the crack growth becomes more regular, allows for a more consistent and less noise-sensitive calculation of the inclinometer-based fracture toughness.

Conversely, for the energy-based and compliance methods, mainly the initial propagation stage was considered, due to limitations in reconstructing the crack evolution. As shown in Figure 3.17, only the first crack fronts are clearly identifiable, while the subsequent regions, characterized by fibre breakage and damage, are less readable for quantitative analysis. It should be noted that the different methods were applied to distinct stages of crack propagation. Consequently, the discrepancies observed between the calculated values of G_{Ic} are not solely related to the mathematical formulation of each method, but also to the evolution of the fracture resistance during crack growth.

Since the experimental results suggest a rising R-curve behaviour associated with fibre bridging mechanisms, the fracture toughness cannot be considered strictly constant throughout propagation. Methods applied at different crack lengths therefore capture different stages of the resistance curve, which naturally leads to variations in the estimated G_{Ic} values.

The fracture surface of Test 3, shown in Figure 3.18, appears noticeably lighter compared to the other specimens, suggesting a limited presence of fibre bridging during crack propagation. Fibre bridging occurs when some fibres do not fail immediately as the crack advances but remain partially intact, spanning across the crack faces and transmitting tensile stresses between them. This mechanism introduces additional energy dissipation and crack-closure effects, leading to an increase in the apparent fracture toughness.

In the case of Test 3, the reduced evidence of fibre bridging implies that the measured fracture toughness is less influenced by additional toughening mechanisms. Accordingly, the value of G_{Ic} obtained from the inclinometer-based method is 0.27 N/mm, which is reasonable and consistent with the order of magnitude provided by the other evaluation methods.

In light of these observations, and considering the absence of a clear stabilization of the G_{Ic} curve, the inclinometer-based estimate cannot be regarded as fully reliable. Nevertheless, it is reported for completeness, while the energy-based and compliance methods, being based on global energetic quantities, provide more stable and consistent results.

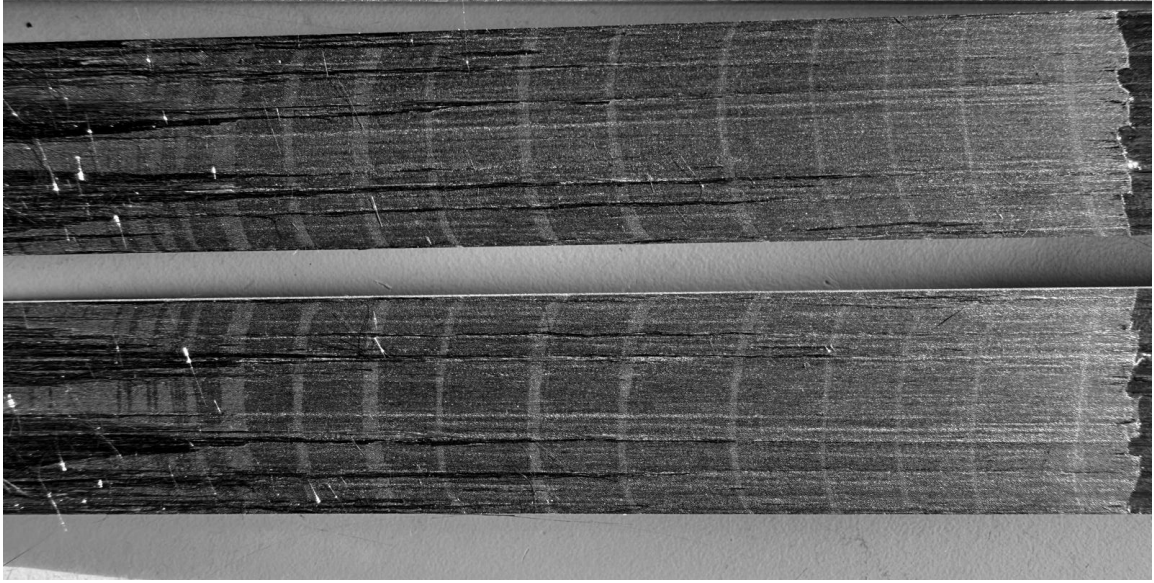


Figure 3.18: Fracture surface of the DCB specimen Test 3.

3.5 End-Notched Flexure (ENF) — Mode II Interlaminar Fracture Toughness

3.5.1 Specimen and Experimental Setup

The objective of the ENF test is the determination of the critical Mode II fracture toughness, G_{IIc} .

The specimens were cut from the same laminate panel used for the DCB tests, namely a $[0]_{32}$ configuration. The main difference lies in the initial crack length: each ENF specimen was prepared with a different initial crack length a_0 in order to investigate its influence on stiffness and fracture behaviour.

The geometrical dimensions of the specimens are reported in Figure 3.19 and summarised in Table 3.7. A total of five specimens were tested.

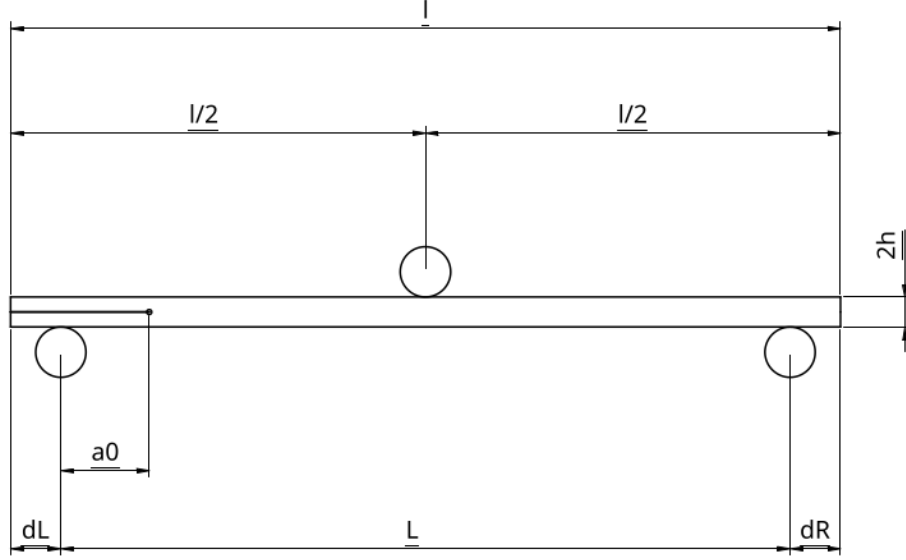


Figure 3.19: Geometrical scheme of the End-Notched Flexure.

Table 3.7: ENF dimensions

Test	L [mm]	B [mm]	h [mm]	l [mm]	a_0 [mm]	d_L [mm]	d_R [mm]
1	145	20	3	165	25	5	15
2	145	20	3	165	35	5	15
3	145	20	3	165	45	5	15
4	145	20	3	165	55	5	15
5	145	20	3	165	65	5	15

In order to improve test stability, the lower support pin located on the pre-cracked side was positioned closer to the specimen edge compared to the opposite support pin. This asymmetric configuration was adopted to promote more stable crack propagation.

During the test, a maximum displacement of -25 mm was imposed at a constant crosshead speed of 5 mm/min. The applied load and the corresponding vertical displacement were measured through the internal sensors of the testing machine.

3.5.2 Data Reduction and Evaluation of G_{IIc}

The Mode II fracture toughness was evaluated using two different approaches, both derived from the theoretical framework described in Section 2.5.3.

- **Beam Theory approach:** the critical value of G_{IIc} was calculated directly from the analytical expression derived under linear elastic beam assumptions, reported in Equation 2.19.
- **Compliance method:** the experimental compliance values were first determined from the force–displacement curves and correlated with the crack length according to the polynomial calibration law $C(a) = ma^3 + c_0$, introduced in Equation 2.20.

The fracture toughness was then obtained from the energetic relation $G_{IIc} = \frac{F^2}{2B} \frac{dC}{da}$, as reported in Equation 2.3.

By differentiating the calibration law, $\frac{dC}{da} = 3ma^2$, the final expression used in this work becomes:

$$G_{IIc} = \frac{3mF^2a^2}{2B} \quad (3.6)$$

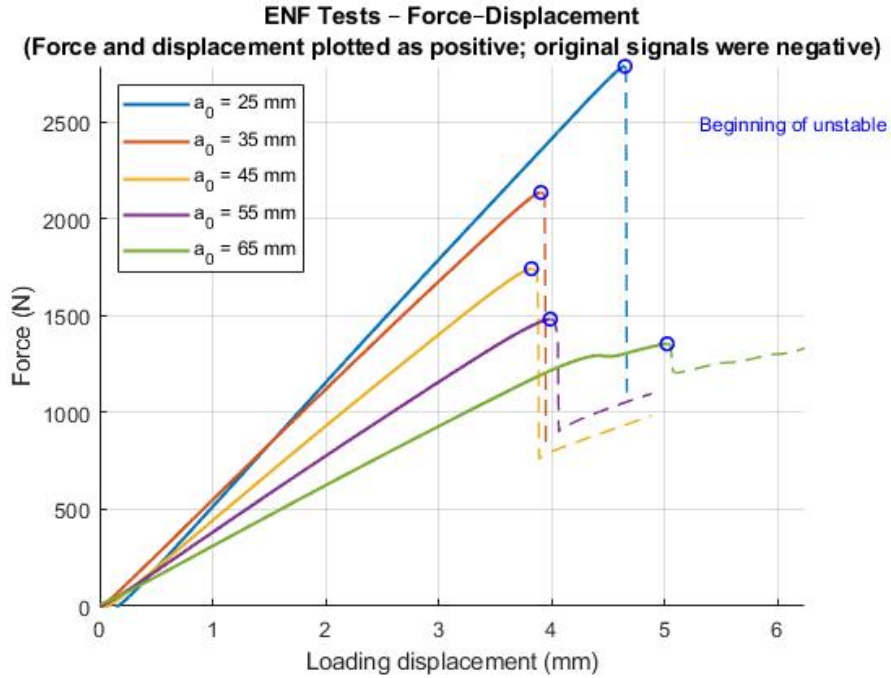


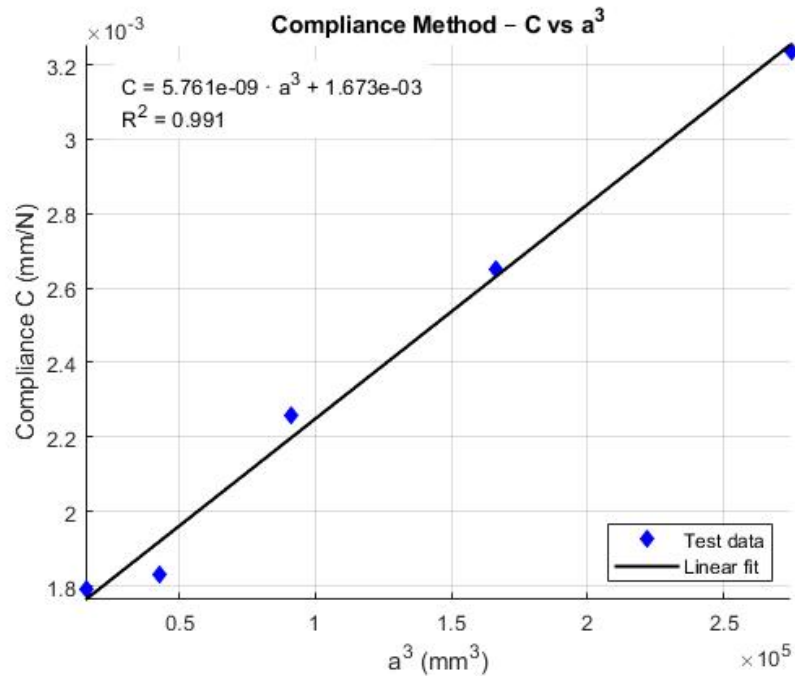
Figure 3.20: ENF:Force vs Displacement.

3.5.3 Results and Discussion

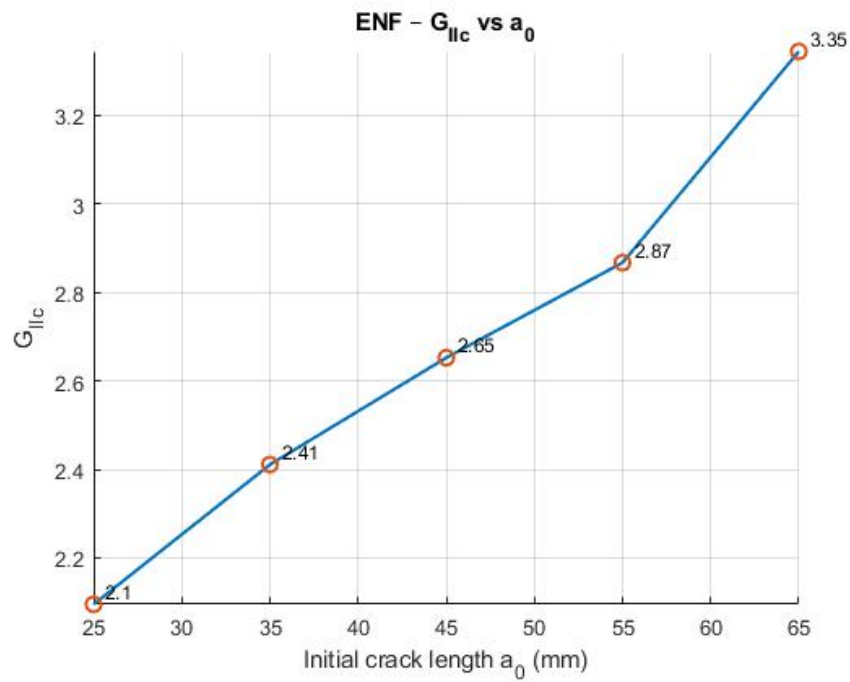
Figure 3.20 reports the complete set of load–displacement curves for all tested specimens. A clear difference in behaviour can be observed as a function of the initial crack length. In particular, the specimen stiffness is

strongly influenced by a_0 , decreasing as the initial crack length increases. Moreover, crack propagation exhibits a pronounced instability, highlighted by the sudden load drop occurring at failure.

The results obtained using the compliance method are shown in Figure 3.21b. In particular, the linear relationship between compliance C and a^3 is reported, together with the evolution of the Mode II fracture toughness as a function of the initial crack length a_0 .



(a) Compliance method: representation of C as a function of a^3 and regression used for calibration.



(b) Compliance method: representation of G_{IIC} as a function of a .

As expected, compliance, being the inverse of stiffness, increases with

increasing crack length. Furthermore, the critical Mode II fracture toughness G_{IIc} shows a tendency to increase with a_0 .

The final values obtained from the experimental campaign are summarised in Table 3.8.

Table 3.8: ENF test results

Test	Peak Force[N]	G_{IIc} [N/mm] Compliance	G_{IIc} [N/mm] Beam theory
Test 1	2786	2.0952	2.2215
Test 2	2134	2.4110	2.5563
Test 3	1741	2.6526	2.8125
Test 4	1481	2.8676	3.0404
Test 5	1353	3.3452	3.5468

As previously observed, the value of G_{IIc} appears to increase with increasing initial crack length a_0 . However, from a theoretical point of view, such behaviour should not occur: the interlaminar fracture toughness is a material property and, under ideal assumptions, it should remain substantially independent of the initial crack length.

The increase observed in the experimental results can be interpreted by formulating several hypotheses, although a definitive cause cannot be identified with certainty. Possible explanations include the intrinsic instability of the ENF configuration, local contact effects induced by the loading pin, and the asymmetric positioning of the support pins adopted in the present setup.

From the Beam Theory formulation (Equation 2.19), it can be observed that G_{IIc} depends, among all parameters, primarily on two variables, since the others remain constant throughout the test campaign: the peak load F_{peak} and the initial crack length a_0 , both appearing squared in the expression. In order for G_{IIc} to remain approximately constant from one specimen to another, the percentage increase in crack length should be compensated by a proportional percentage decrease in the peak force.

A simple estimation shows that a_0 increases from 25 mm to 65 mm. When considering its contribution in the analytical formulation, this corresponds to a multiplication factor of $\frac{65^2}{25^2} \approx 6.76$, while the peak force decreases from 2786 N to 1353 N, corresponding to a factor of $\frac{1353^2}{2786^2} \approx 0.24$.

The combined effect of these two contributions leads to an overall variation of about $6.76 \times 0.24 \approx 1.6$, which is consistent with the experimentally observed increase in G_{IIc} . Since these variations are not proportional, the contribution related to the crack length tends to dominate the expression of G_{IIc} , leading to the observed increasing trend.

A possible explanation for this behaviour may be related to the adopted

test configuration. In the present setup, the support pins were not symmetrically positioned with respect to the specimen edges, in order to promote improved stability of crack propagation. Although this configuration facilitates testing, it deviates from the ideal symmetric ENF assumptions underlying analytical beam formulations and may introduce a non-uniform stress distribution along the specimen.

In addition, the presence of the central loading pin induces a localized compressive stress field beneath the contact region. As the initial crack length increases, the crack tip approaches the loading point, enhancing the interaction between the delamination front and the compressive stress field generated by the pin. This geometric evolution may alter the local shear stress distribution and increase contact and friction effects between the delaminated surfaces.

Consequently, the assumption of pure Mode II Linear Elastic Fracture Mechanics conditions is not strictly satisfied. The calculated G_{IIc} values may therefore incorporate configuration-dependent contributions associated with contact pressure, friction and mixed-mode effects, rather than representing a purely intrinsic material parameter.

In addition, differences in post-peak behaviour can be observed among the tested specimens. The first two tests (with $a_0 = 25$ mm and $a_0 = 35$ mm) exhibit a sudden and complete load drop immediately after reaching the peak force. This behaviour indicates a rapid unstable crack propagation occurring right after the critical load is attained. It is important to underline that this does not compromise the determination of the critical value itself, since G_{IIc} is evaluated at the peak load, prior to the instability.

The specimens with larger initial crack lengths ($a_0 = 55$ mm and $a_0 = 65$ mm) show a less pronounced load drop and a comparatively smoother post-peak evolution. In these cases, crack propagation appears slightly more gradual after the peak force is reached, although the intrinsic instability of the ENF configuration remains present.

For this reason, and considering the more regular post-peak response observed for the specimens with $a_0 = 45$, 55 and 65 mm, a representative reference value of G_{IIc} was computed as the average of the last three tests, leading to a mean value of approximately $G_{IIc} \approx 3$ N/mm.

3.6 Four-Point End-Notched Flexure (4ENF) — Mode II Interlaminar Fracture Toughness

3.6.1 Specimen and Experimental Setup

The experimental campaign was carried out following the theoretical framework presented in Section 2.5.4. The specimens were cut from the same laminate panel used for the DCB and ENF tests. The geometrical dimensions

and the configuration of the experimental setup are reported in Figure 4.33 and summarized in Table 3.9.

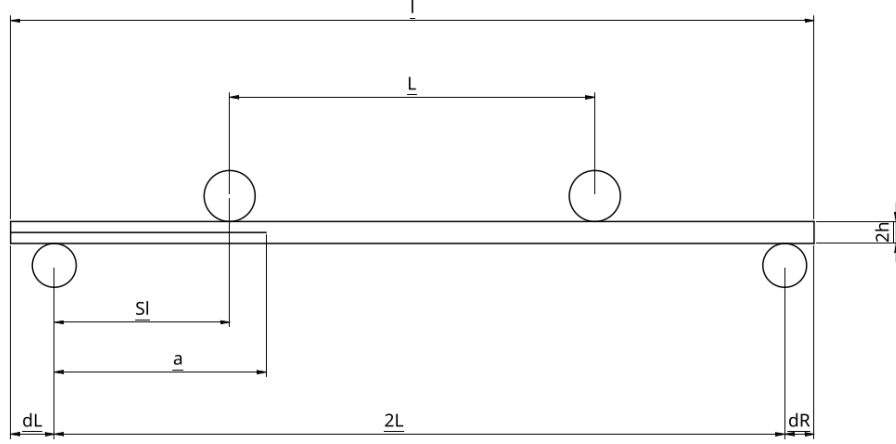


Figure 3.22: Geometrical scheme of the 4End-Notched Flexure.

Table 3.9: 4ENF dimensions

Test	L [mm]	B [mm]	h [mm]	l [mm]	a_0 [mm]	d_L [mm]	d_R [mm]	S_l [mm]
1	100	20	3	220	63	8	12	52
2	100	20	3	220	68	3	17	52
3	100	20	3	220	68	3	17	52

A maximum displacement of -15 mm was imposed at a constant crosshead speed of 2 mm/min. The applied load F and the corresponding vertical displacement were recorded through the internal sensors of the testing machine.

3.6.2 Data Reduction and Evaluation of G_{IIc}

The Mode II fracture toughness G_{IIc} was evaluated using the two analytical formulations introduced in the theoretical section:

- **Beam theory formulation:** G_{IIc} was calculated using the following expression:

$$G_{IIc} = \frac{9F^2L^2}{64EB^2h^3} \quad (3.7)$$

which is derived from the classical beam theory formulation for the 4ENF configuration.

- **Friction-corrected formulation:** G_{IIc} was calculated using Equation 2.24, which accounts for the contribution of interlaminar friction through the friction coefficient α .

Figure 3.23 shows the force–displacement curves obtained from the three tested specimens. For the calculation of G_{IIc} , the critical load F was defined as the peak load F_{peak} for both formulations.

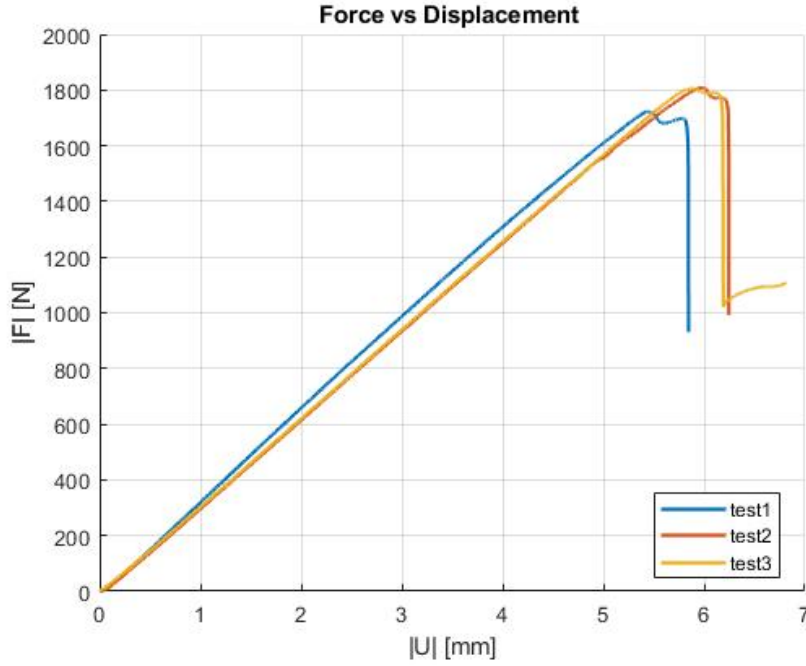


Figure 3.23: 4ENF:Force vs Displacement.

3.6.3 Results and Discussion

From Figure 3.23, it can be observed that the three 4ENF tests exhibit a very similar global behaviour. In particular, the initial elastic stiffness is comparable among all specimens, as well as the peak load values. This similarity is even more evident for Tests 2 and 3, which share the same geometrical configuration. Test 1, on the other hand, shows a slightly different response, which can be explained by the fact that the specimen was shifted by approximately 5 mm with respect to the configuration adopted for Tests 2 and 3.

According to the theoretical framework, the 4ENF configuration is expected to provide improved and more stable crack propagation compared to the classical ENF test under ideal geometric and contact conditions, characterized by a well-developed plateau region after the peak load.. However, in the present experimental campaign, although a plateau region is visible, it

appears limited in extension compared to the ideal behaviour typically described in literature. The post-peak response shows a relatively short stable region, followed by a sudden instability identified by a sharp load drop.

This behavior indicates that, while the 4ENF configuration improves stability compared to the classical ENF test, the stable propagation regime remained limited in extension in the present setup. Possible reasons may include friction between the crack surfaces, slight asymmetries in the experimental setup, or a non-optimal initial crack length.

The G_{IIc} results reported in Table 3.10 show good consistency among the three tests, particularly for Tests 2 and 3, as expected from their identical configuration. Since the friction coefficient between the crack surfaces was unknown, a sensitivity analysis was performed by varying the parameter α . The results show that an increase in the friction coefficient leads to a reduction of the computed G_{IIc} value. This can be interpreted as a deterioration of the apparent fracture resistance, since part of the measured load is associated with frictional effects rather than purely with crack propagation resistance.

Table 3.10: 4ENF experimental results.

Test	$G_{IIc}(\alpha = 0.15)$	$G_{IIc}(\alpha = 0.25)$	$G_{IIc}(\alpha = 0.35)$	$G_{IIc}(\alpha = 0.45)$	G_{IIc} Beam Theory
1	3.5693	3.5140	3.4591	3.4046	3.3979
2	3.9346	3.8736	3.8131	3.7530	3.7456
3	3.9187	3.8579	3.7977	3.7378	3.7305
Average	3.8075	3.7485	3.6899	3.6318	3.6247

The values obtained using the beam theory formulation are slightly lower than those obtained with the friction-corrected approach. This difference is likely due to the fact that the classical beam formulation completely neglects the contribution of interlaminar friction.

The G_{IIc} values obtained from the 4ENF tests are comparable with those derived from the last three ENF tests, which were considered representative. The 4ENF values are slightly higher, possibly due to the fact that the peak load was used for the calculation instead of the plateau load, which is marginally lower. Since the plateau region was limited and not clearly defined, the peak load was selected to ensure objective and reproducible identification of the critical load.

Therefore, although the stable regime was limited in extension, the 4ENF results can be considered consistent and physically representative of the Mode II fracture resistance of the material. The good agreement among the three tests and the coherence with the ENF results confirm the reliability of the identified order of magnitude of G_{IIc} .

3.7 Discussion of Experimental Results

In conclusion, the experimental campaign allowed the characterization of the interlaminar fracture behaviour of the investigated composite material under both Mode I and Mode II loading conditions.

For Mode I, the DCB configuration provided consistent fracture toughness values G_{Ic} , with good repeatability among the tested specimens. The force–displacement responses exhibited coherent peak loads and propagation behaviour, confirming the reliability of the adopted experimental procedure.

For Mode II, the ENF tests highlighted a clear dependence of the results on the initial crack length. In particular, shorter initial cracks resulted in more unstable propagation and a higher sensitivity of the peak load to the specific test conditions. This confirms the crucial role of the initial crack length in governing the stability of the ENF configuration.

The 4ENF configuration led to improved stability conditions compared to the classical ENF setup. However, the post-peak plateau region was limited in extension, suggesting that local compressive effects and interfacial friction influence the evolution of delamination in a non-negligible manner.

Despite the limitations observed during testing, the obtained results are coherent and can be interpreted as effective fracture parameters representative of the specific experimental configurations adopted. These experimentally identified values will therefore be used in the subsequent numerical modelling phase for validation of the implemented finite element models.

Chapter 4

Numerical Modelling Results

This chapter presents the results obtained through numerical modelling of the experimental tests previously described. All simulations were performed using the finite element software Abaqus, with the objective of reproducing the mechanical response observed during laboratory testing and providing a direct comparison between numerical predictions and experimental data.

Interlaminar fracture behaviour was investigated using two distinct modelling strategies: the Virtual Crack Closure Technique (VCCT) and a surface-based cohesive contact formulation. The adoption of both approaches allows a comparative assessment of their capability to predict crack initiation, propagation behaviour, and energy release rates under the different loading configurations considered.

The composite laminate was modelled as a linear elastic orthotropic material defined by its engineering constants. The mechanical properties adopted in the simulations are summarized in Table 4.1 and were used consistently across all numerical analyses, including tensile $\pm 60^\circ$, three-point bending, DCB, ENF and 4ENF configurations.

Table 4.1: Engineering constants used for the numerical simulations.

Property	Symbol	Value
Effective flexural modulus	E_f	113697 N/mm ²
Longitudinal Young's modulus	E_1	135000 N/mm ²
Transverse Young's modulus	E_2	8500 N/mm ²
Out-of-plane Young's modulus	E_3	8500 N/mm ²
In-plane shear modulus	G_{12}	4200 N/mm ²
Transverse shear modulus	G_{13}	4200 N/mm ²
Out-of-plane shear modulus	G_{23}	3200 N/mm ²
Poisson's ratio	ν_{12}	0.33
Poisson's ratio	ν_{13}	0.33
Poisson's ratio	ν_{23}	0.45

The effective flexural modulus E_f was adopted in selected simulations in order to compensate for the increased stiffness introduced by the plane strain formulation, thus providing a more representative global bending response compared to experimental observations.

4.1 Interlaminar Fracture Modelling Approach

4.1.1 Virtual Crack Closure Technique (VCCT)

Within the VCCT framework, crack propagation is governed by an energy-based criterion. The energy release rates G_I , G_{II} and G_{III} are computed at the crack tip and compared to the corresponding critical values. Mixed-mode interaction was defined using the Benzeggagh–Kenane (BK) law, allowing the effective critical energy release rate to depend on the mode mixity.

The interaction property adopted in Abaqus includes:

- Tangential behaviour: frictionless
- Normal behaviour: hard contact
- Fracture criterion type: VCCT (energy-based formulation)
- Mixed-mode behaviour: Benzeggagh–Kenane law
- Critical energy release rates G_{Ic} , G_{IIc} and G_{IIIc}

The crack growth direction relative to the local interface system was defined according to the maximum tangential stress criterion. In the present study, crack propagation is constrained along the predefined interface; therefore, the selected directional criterion does not influence the crack path but ensures consistency within the VCCT implementation.

The tolerance for unstable crack propagation was left at the default Abaqus setting. This internal control limits excessive crack advancement within a single increment and contributes to numerical stability during post-peak propagation. A small viscosity parameter was introduced to improve convergence and mitigate potential numerical instabilities during unstable crack growth.

4.1.2 Surface-Based Cohesive Contact

For the cohesive contact approach, interfacial behaviour was described through a traction–separation law. The interaction property included:

- Tangential behaviour: frictionless
- Normal behaviour: hard contact

- Cohesive behaviour defined by penalty stiffness parameters K_{nn} , K_{ss} and K_{tt}
- Damage initiation criterion
- Damage evolution law based on fracture energy

The interface initially behaves elastically according to the specified stiffness parameters. Once the damage initiation criterion is satisfied, stiffness degradation evolves progressively according to the prescribed fracture energy, leading to complete decohesion and crack propagation along the interface.

4.2 Three-Point Bending (3PB)

4.2.1 Numerical Setup

In order to simulate the three-point bending test, a two-dimensional modelling approach was adopted by neglecting the specimen width and representing only its longitudinal section. This modelling strategy allows the evaluation of the structural response per unit width, while the out-of-plane dimension is considered when scaling numerical reaction forces for comparison with experimental data.

The numerical model consists of a deformable planar part representing the longitudinal section of the specimen and analytically rigid planar surfaces modelling the lower support pins and the upper loading pin. Figure 4.1 shows a representative view of the numerical model geometry.

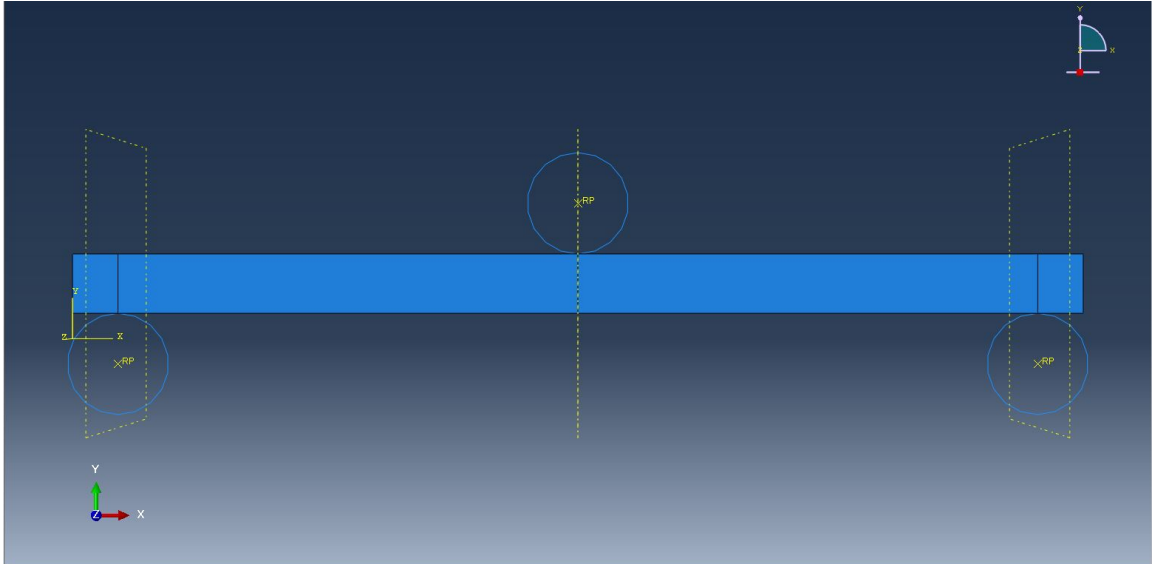


Figure 4.1: Finite element model of the three-point bending specimen.

The material properties are those reported in Table 4.1.

Contact interactions were defined between the rigid pins and the specimen surfaces in order to prevent interpenetration during loading. The normal behaviour was defined as hard contact, while tangential behaviour was assumed frictionless, since friction effects were considered negligible for the present configuration.

In the initial step, the necessary translational degrees of freedom were constrained in order to prevent rigid body motion. In the subsequent loading step, the two lower support pins were fully constrained, while the upper loading pin was constrained in the horizontal direction and subjected to a prescribed vertical displacement in the negative y direction, reproducing the experimental bending configuration.

The boundary conditions adopted in the model are illustrated in Figure 4.2.

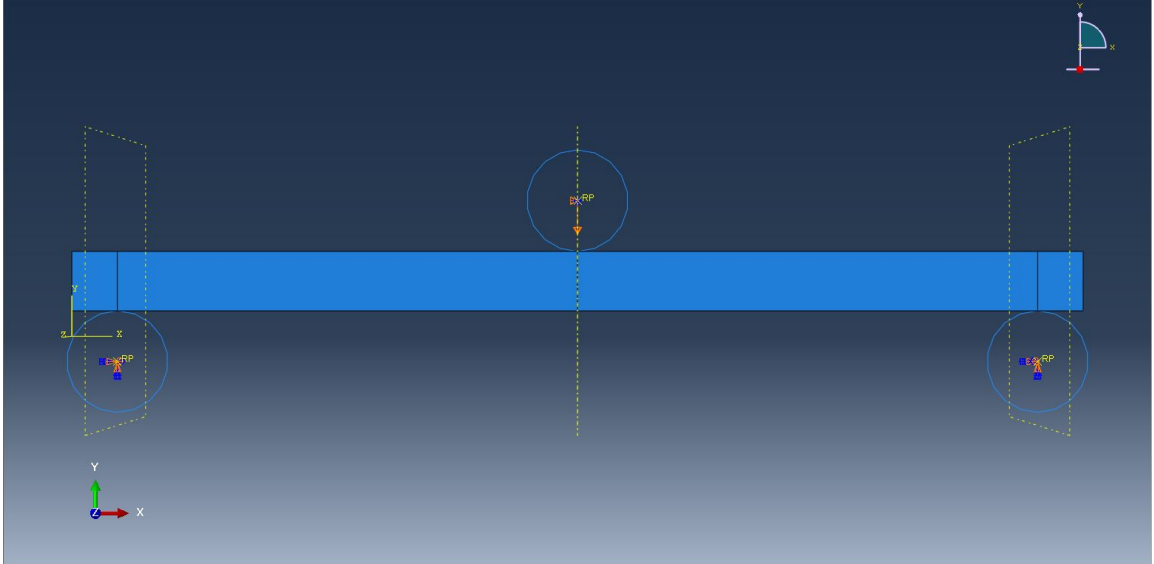


Figure 4.2: Boundary conditions and loading configuration of the three-point bending model.

For the three-point bending configuration, a uniform mesh size of 0.5 mm was adopted over the entire specimen, as shown in Figure 4.3. The selected mesh size represents a compromise between simulation accuracy and computational cost.

A structured quadrilateral mesh was employed. The elements belong to the plane strain family and were defined as CPE4 elements (four-node bilinear plane strain quadrilateral elements), as visible in Figure 4.3.

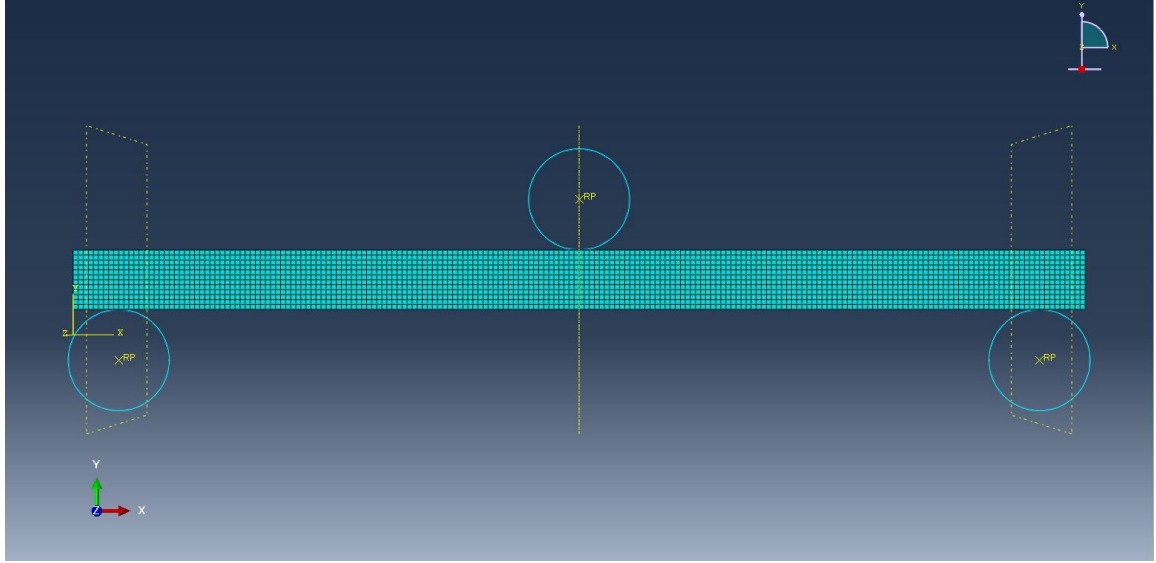


Figure 4.3: Finite element mesh of the three-point bending model.

4.2.2 Results and Comparison

Figure 4.4 shows the deformed configuration of the specimen after the application of the load. As expected for a three-point bending configuration under symmetric boundary conditions, the deformation is symmetric with respect to the mid-span of the specimen. The maximum vertical displacement occurs at the centre of the beam, where the load is applied.

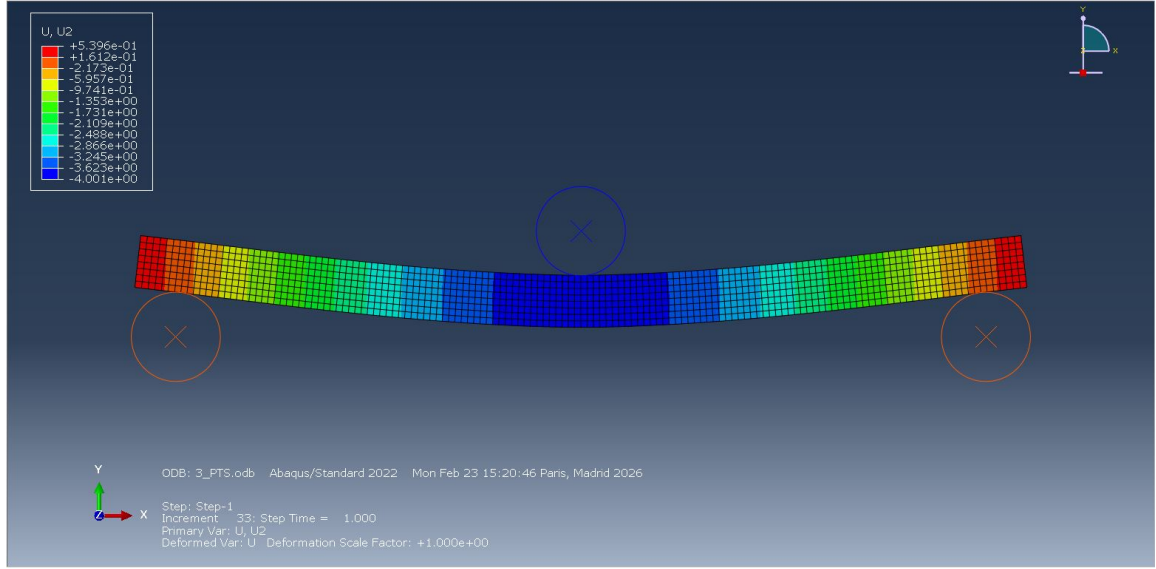


Figure 4.4: Deformed configuration of the three-point bending specimen (vertical displacement U_2).

Figure 4.5 represents the longitudinal normal stress component S_{11} resulting from the applied load. As expected from classical beam theory, the upper surface of the specimen is subjected to compressive stresses, while the lower surface is subjected to tensile stresses. In the central region of the beam, the stress distribution varies approximately linearly through the thickness, transitioning from compression at the top to tension at the bottom, in agreement with Euler–Bernoulli beam theory.

It can be observed that the S_{11} distribution is not perfectly symmetric, with slightly higher compressive stresses in the upper region of the specimen. This behaviour is likely associated with the local compressive effects induced by the contact with the loading pin.

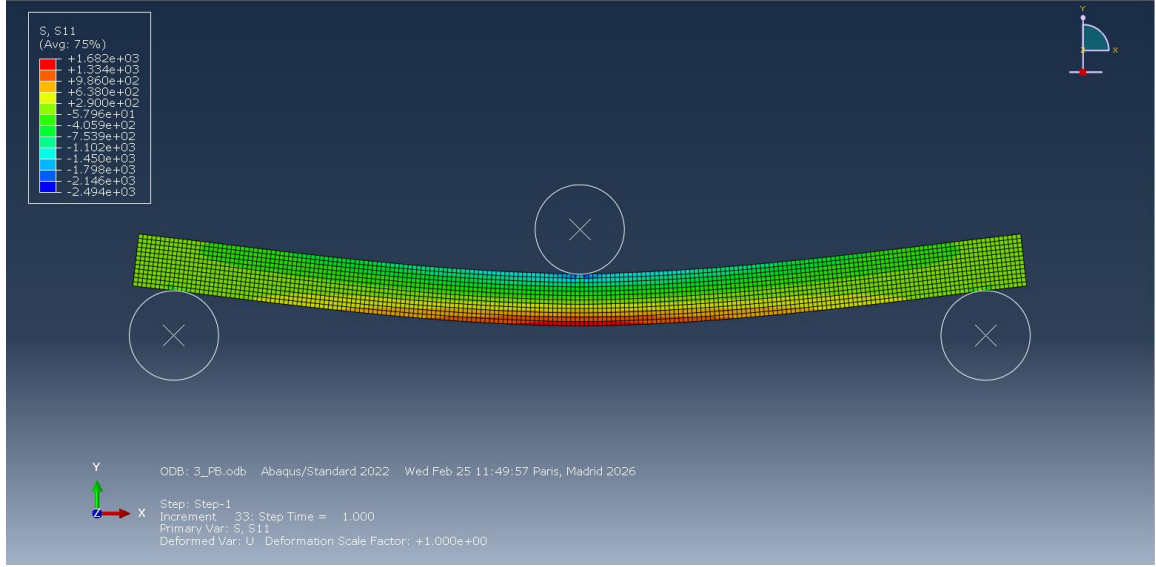


Figure 4.5: Distribution of longitudinal normal stress σ_{11} in the three-point bending specimen.

The local compressive effect generated by the loading pin is more clearly visible in Figure 4.6, where the transverse normal stress component S_{22} is reported. A localized compressive stress concentration can be observed directly beneath the loading pin, confirming the presence of contact-induced stresses superimposed on the global bending behaviour. The higher compressive state observed in S_{22} beneath the loading pin is attributed to the local contact pressure transmitted by the rigid indenter. This compressive stress is not related to the global bending behaviour, but rather to the localized contact interaction between the pin and the specimen surface. Away from the loading point, the transverse normal stress decreases significantly, confirming that the global response remains dominated by bending stresses.

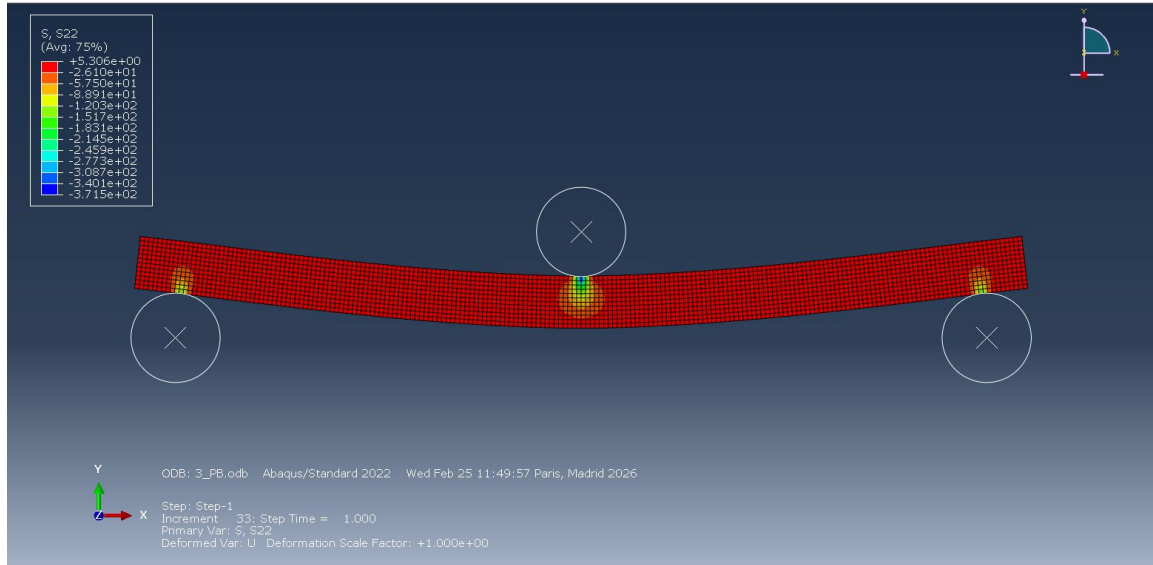


Figure 4.6: Distribution of transverse normal stress σ_{22} .

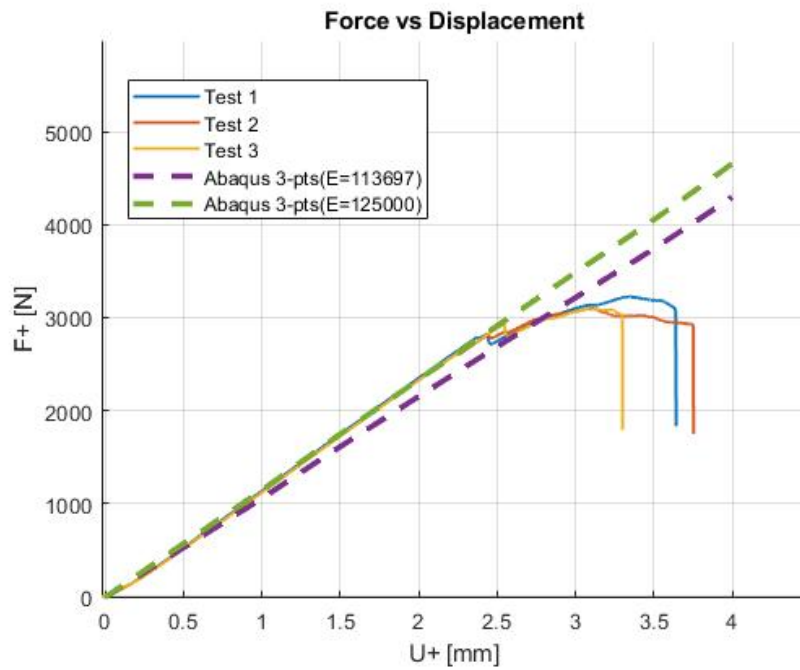


Figure 4.7: Comparison between experimental and numerical force–displacement curves for the three-point bending test.

Figure 4.7 shows the comparison between the experimental and numerical force–displacement curves obtained from the three-point bending test.

When the experimentally identified Young’s modulus ($E = 113697 \text{ N/mm}^2$) is introduced into the numerical model, the predicted initial slope is slightly lower than the experimental one. In order to obtain a better agreement in terms of stiffness, a higher modulus value of $E = 125000 \text{ N/mm}^2$ was adopted, leading to a closer match with the experimental response.

The observed discrepancy may be attributed to a combination of experimental uncertainties in the identification of the elastic modulus and modelling idealizations adopted in the numerical framework. In particular, the flexural modulus obtained from the three-point bending test is evaluated using a relation in which the span length L appears to the third power. As a consequence, even small uncertainties in the measurement of the span length or slight misalignments in the positioning of the experimental setup can produce noticeable variations in the estimated value of the flexural modulus. Such effects may therefore contribute to the difference observed between the experimentally derived modulus and the value that best reproduces the numerical stiffness.

In addition, modelling assumptions such as the plane strain formulation and the hypothesis of perfectly rigid pins may influence the global stiffness predicted by the model. While some of these idealizations can slightly increase the apparent stiffness of the numerical response, other experimental factors such as machine compliance, boundary conditions, or parameter identification procedures may lead to deviations in the opposite direction. The overall difference in slope therefore results from the combined influence of these effects.

Nevertheless, the overall linear elastic behaviour predicted numerically remains consistent with the experimental trend. It should also be noted that no damage or failure model was implemented in the present numerical analysis. The simulation is therefore limited to the elastic regime and does not aim to reproduce material degradation or ultimate failure mechanisms, but rather to validate the global bending stiffness of the laminate.

4.3 Tensile Test ($\pm 60^\circ$)

4.3.1 Numerical Setup

The tensile test was modelled using a shell-based two-dimensional formulation. The specimen was represented by its in-plane geometry, while the laminate thickness and stacking sequence were defined within the shell section properties. This modelling strategy allows the evaluation of the global in-plane mechanical response while fully accounting for the laminate configuration through the assigned ply orientations and thicknesses. The numerical model consists of a single deformable planar part representing the specimen geometry, as shown in Figure 4.8.



Figure 4.8: Finite element model of the tensile specimen.

The composite layup was assigned directly to the planar part using the laminate stacking sequence. The ply orientations and stacking configuration are illustrated in Figure 4.9. The laminate definition was introduced through a shell section, specifying the material properties, thickness, and orientation of each ply.

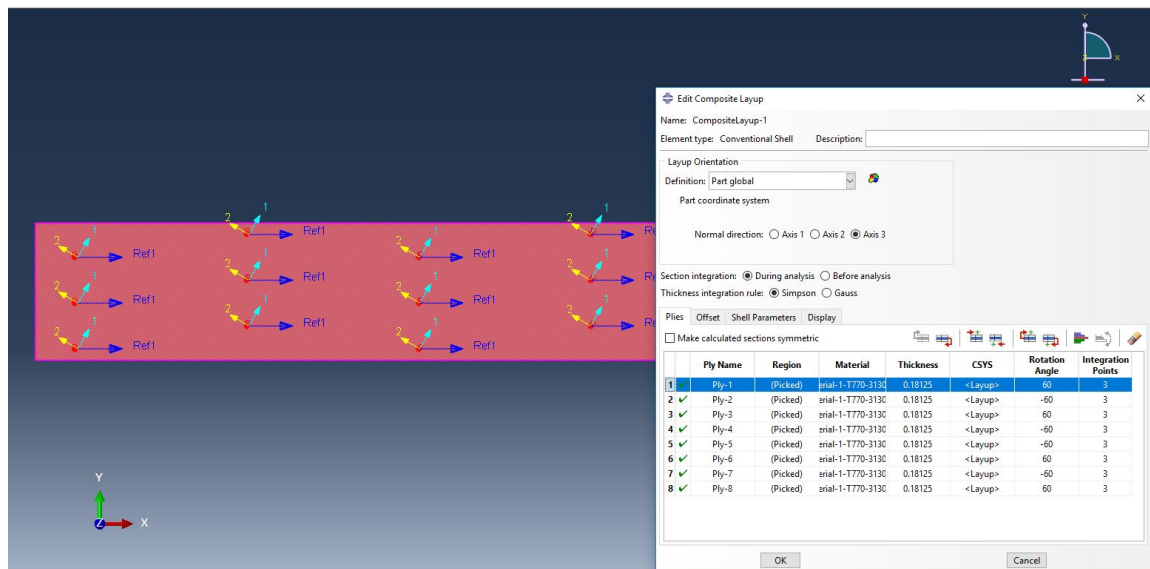


Figure 4.9: Composite layup definition of the tensile specimen.

In order to reproduce the tensile loading configuration, a reference point

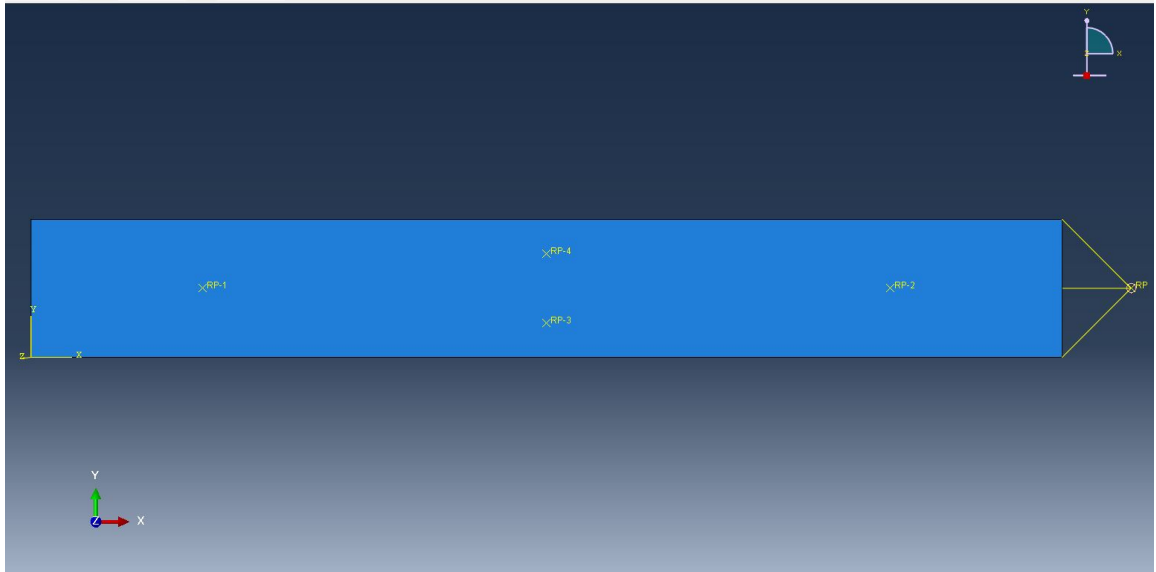
was created and kinematically coupled to the loaded edge of the specimen. The opposite edge was fully constrained to reproduce the clamped condition applied during the experimental test.

In the initial step, all translational degrees of freedom of the reference point were constrained to prevent rigid body motion. In the subsequent loading step, a prescribed displacement was applied to the reference point along the longitudinal direction of the specimen.

The boundary conditions and coupling interaction adopted in the model are illustrated in Figure [4.10](#).



(a) Boundary conditions and loading configuration.



(b) Kinematic coupling between the reference point and the loaded edge.

Figure 4.10: Numerical setup of the tensile test model: boundary conditions and interaction definition.

A uniform mesh size of 0.5 mm was adopted over the entire specimen surface. The selected mesh size represents a compromise between simulation accuracy and computational efficiency.

A structured quadrilateral mesh was employed. The elements belong to the shell family and were defined as S4 elements (four-node quadrilateral

shell elements with full integration), as shown in Figure 4.11. The S4 formulation allows the representation of both membrane and bending behaviour while accounting for the laminate stacking sequence directly within the shell section definition.

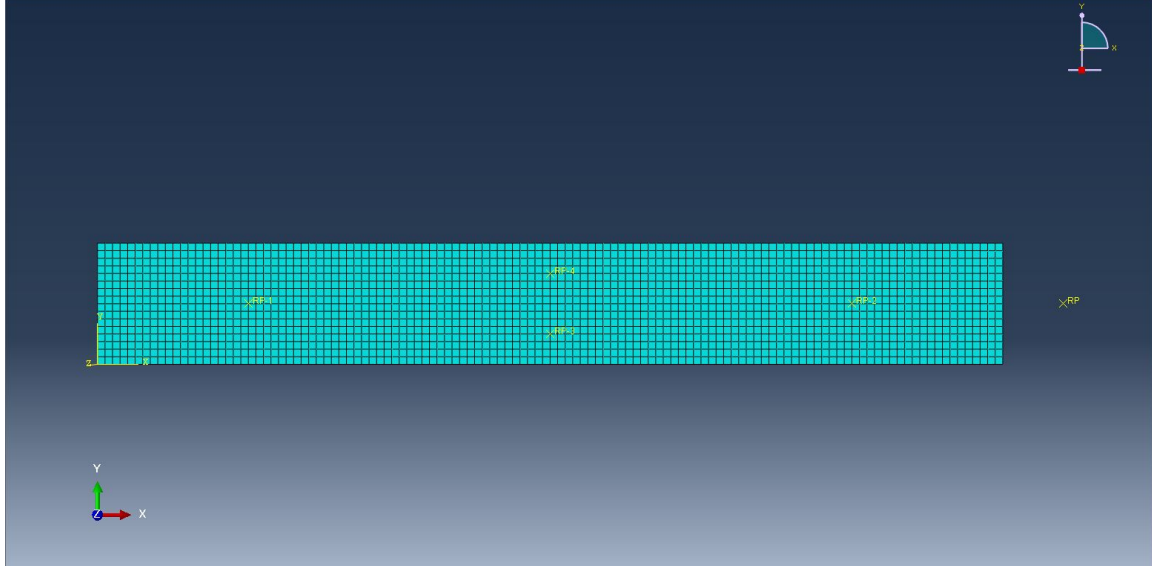


Figure 4.11: Finite element mesh of the tensile specimen.

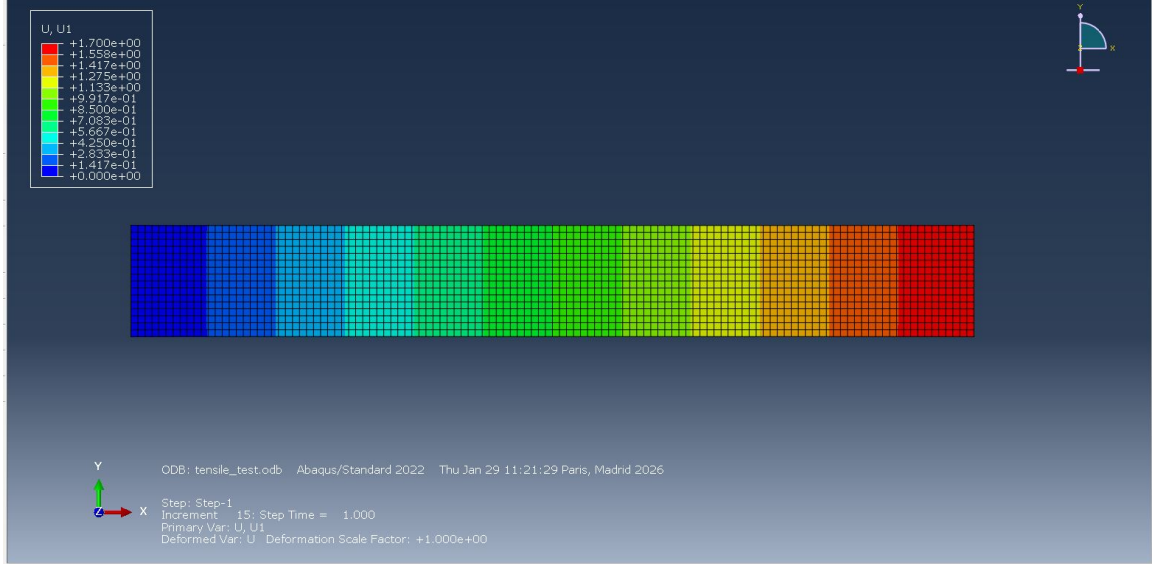
4.3.2 Results and Comparison

The numerical response of the tensile specimen is summarized in Figures 4.12 and 4.13, which report the in-plane displacement fields and the corresponding stress distributions.

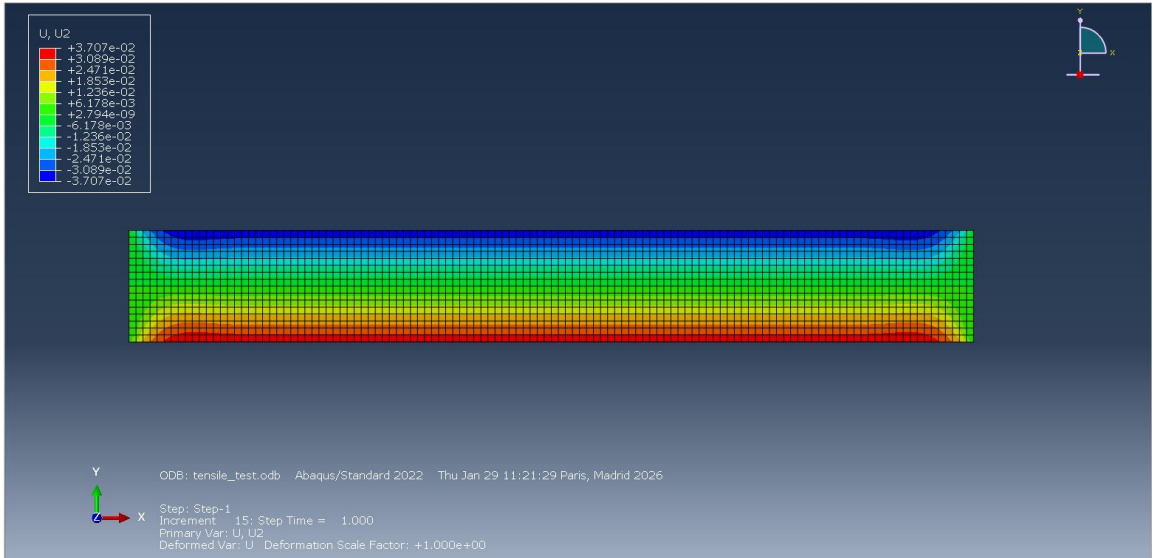
Figure 4.12 shows the longitudinal displacement component U_1 and the transverse displacement component U_2 . The U_1 field exhibits an almost monotonic increase from the constrained side towards the loaded side, indicating that the applied boundary conditions successfully reproduce a predominantly axial loading condition. In the central region of the specimen, the displacement gradient is approximately uniform, which is consistent with a global tensile deformation and limited rigid-body motion.

The U_2 field highlights the transverse deformation of the specimen during loading. A clear sign change across the mid-height of the coupon can be observed, which is coherent with a lateral contraction/expansion response induced by stress transformation effects in the off-axis plies and by the anisotropic behaviour of the $\pm 60^\circ$ stacking sequence. Although the laminate is symmetric and therefore does not exhibit bending-extension coupling, the off-axis orientation of the plies generates in-plane shear and transverse deformation under axial loading. Compared with U_1 , the magnitude of U_2 remains significantly smaller, confirming that the global deformation is

dominated by the imposed axial extension, while transverse effects remain secondary but not negligible.



(a) Longitudinal displacement U_1 .



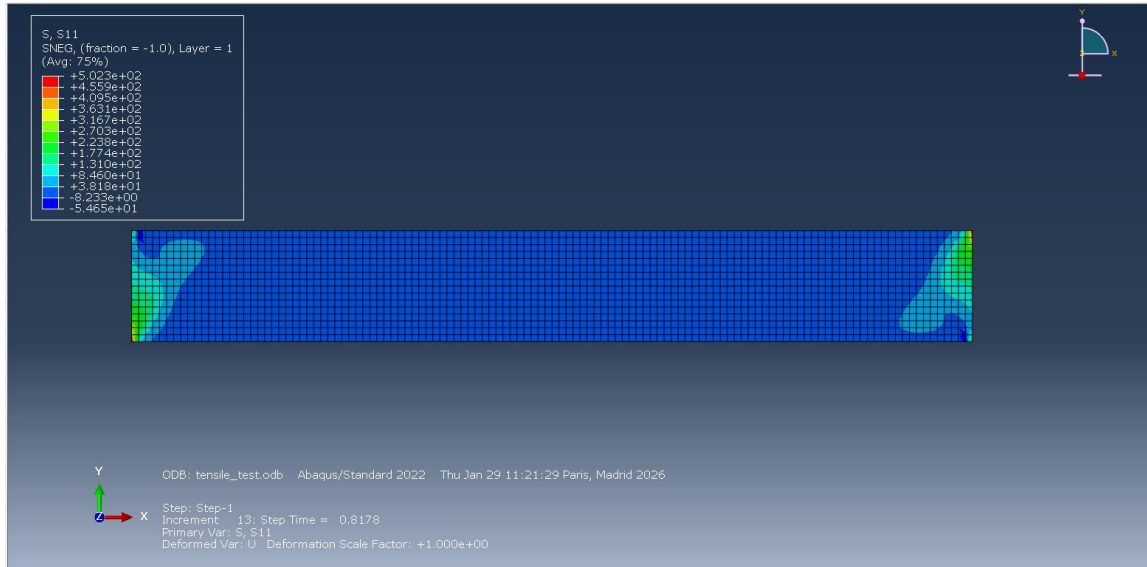
(b) Transverse displacement U_2 .

Figure 4.12: In-plane displacement fields of the tensile specimen.

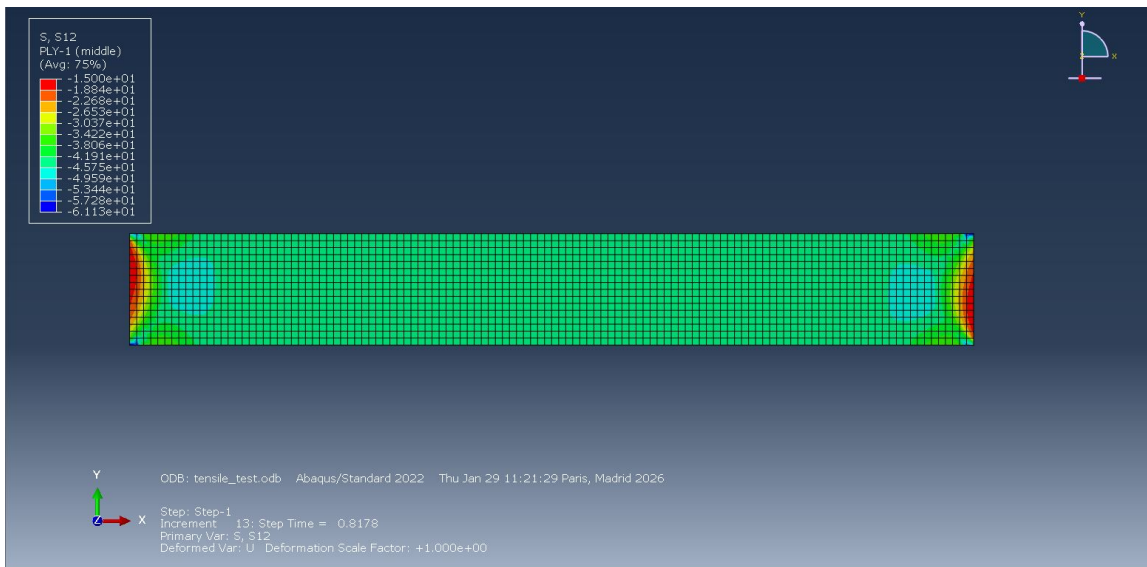
The stress distributions are reported in Figure 4.13. The longitudinal normal stress component S_{11} is predominantly uniform in the gauge region, indicating that the specimen experiences a largely homogeneous ten-

sile stress state away from the boundaries. Higher stress gradients appear in the proximity of the constrained and loaded edges, where the load transfer is enforced through the coupling constraint and the fixed boundary condition. These localized effects are expected and can be interpreted as boundary-condition-induced stress concentrations, which progressively decay when moving towards the central region of the specimen.

The shear stress component S_{12} provides additional insight into the deformation mechanism of the laminate. Unlike a unidirectional 0° coupon, a $\pm 60^\circ$ stacking sequence is expected to develop significant in-plane shear under axial loading, due to the strong coupling between normal and shear components in off-axis plies. Accordingly, the S_{12} field shows non-negligible shear stresses, with the highest values localized near the specimen ends where the load is introduced and redistributed within the laminate. In the central region, the shear stress level remains comparatively lower and more uniform, suggesting that the dominant load transfer occurs primarily at the boundaries, while the gauge section is mainly characterized by a stable combined axial-shear state.



(a) Longitudinal normal stress σ_{11} .



(b) In-plane shear stress σ_{12} .

Figure 4.13: Stress distribution in the tensile specimen.

Finally, Figure 4.14 presents the comparison between the experimental force–displacement curves and the numerical response obtained from the finite element model.

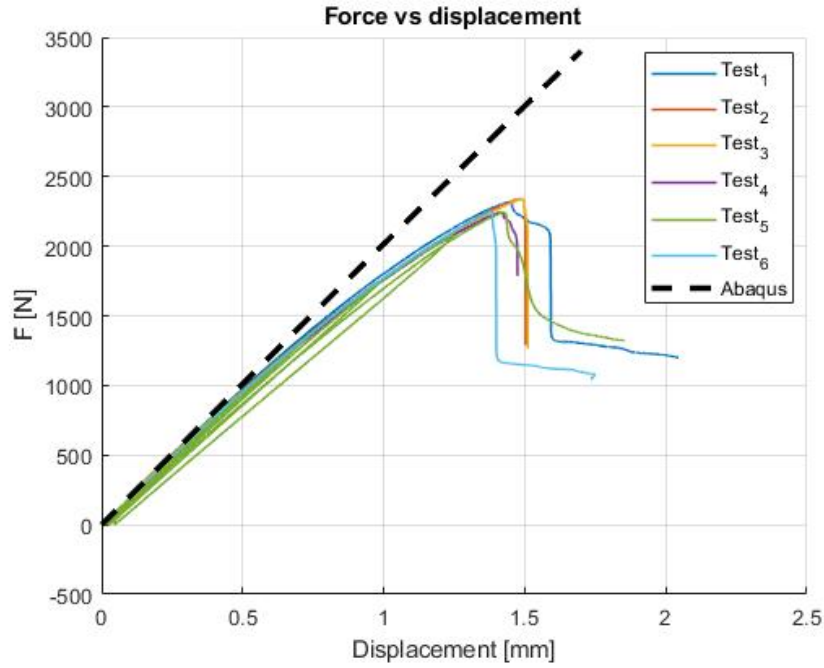


Figure 4.14: Comparison between experimental and numerical force–displacement curves for the tensile test.

As can be observed, the numerical model accurately reproduces the initial linear portion of the experimental curves. The slope predicted by Abaqus is consistent with the experimentally measured stiffness in the elastic regime, confirming that the material properties and laminate definition implemented in the model correctly capture the global elastic behaviour of the specimen.

As the experimental curves approach the maximum load, further differences become evident. The experimental specimens exhibit stiffness degradation and load drops associated with progressive damage mechanisms. The numerical response, instead, remains linear since no failure or damage model was implemented in the present simulation. The objective of the numerical model was the validation of the elastic response of the laminate rather than the reproduction of progressive damage behaviour.

Therefore, the agreement observed in the elastic region can be considered satisfactory with respect to the scope of the numerical analysis.

4.4 Double Cantilever Beam (DCB)

4.4.1 Numerical Setup

The DCB specimen was modelled using a two-dimensional approach by neglecting the specimen width and representing only its longitudinal section. This modelling strategy allows the evaluation of the structural response per unit width, while the out-of-plane dimension is considered when scaling numerical reaction forces for comparison with experimental data.

The numerical model consists of two deformable planar parts representing the upper and lower arms of the specimen. Figure 4.15 shows a representative view of the numerical model geometry.

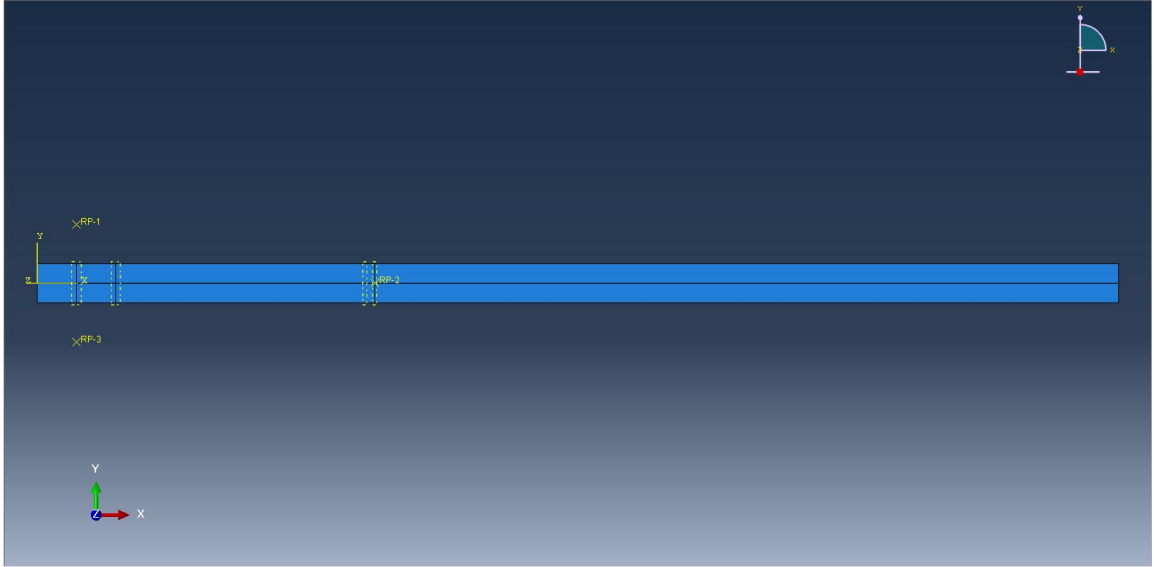


Figure 4.15: Finite element model of the DCB specimen.

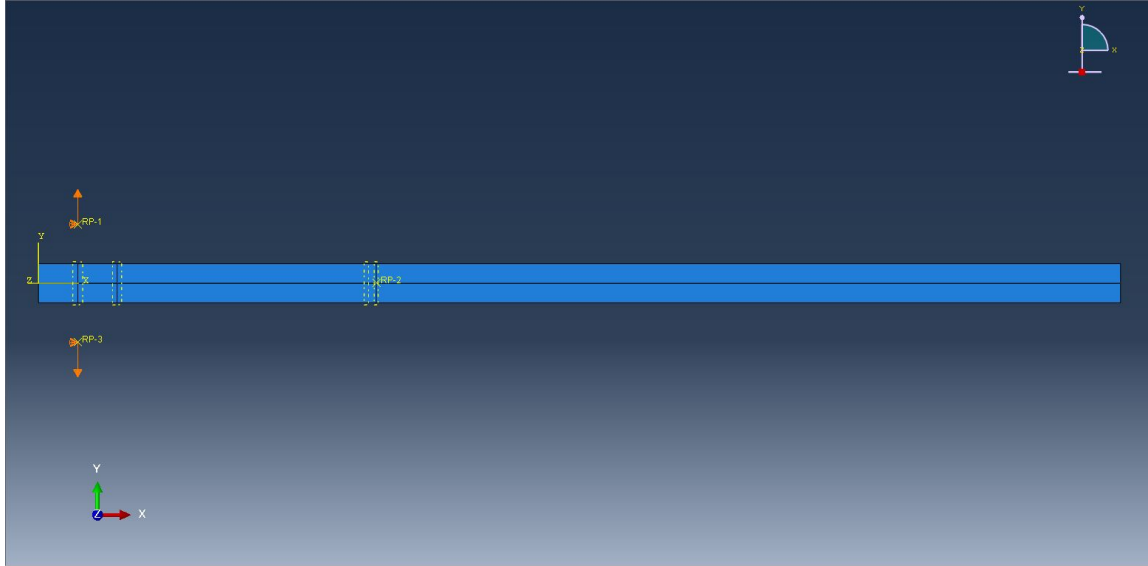
To reproduce the loading hinges used in the experimental setup, two reference points were created, each corresponding to the centre of a loading block. A kinematic coupling constraint was defined between each reference point and the corresponding loading surface of the upper and lower arms.

In the initial step, the two reference points were fully constrained (translations in x and y set to zero) in order to prevent rigid body motion and ensure a stable contact initialization. In the subsequent loading step, the vertical translation U_2 was released and replaced by a prescribed displacement of equal magnitude and opposite sign on the two reference points, while the horizontal translation U_1 was kept constrained. The rotation about the out-of-plane axis was left free to reproduce the hinge behaviour and allow the arms to rotate naturally during crack opening.

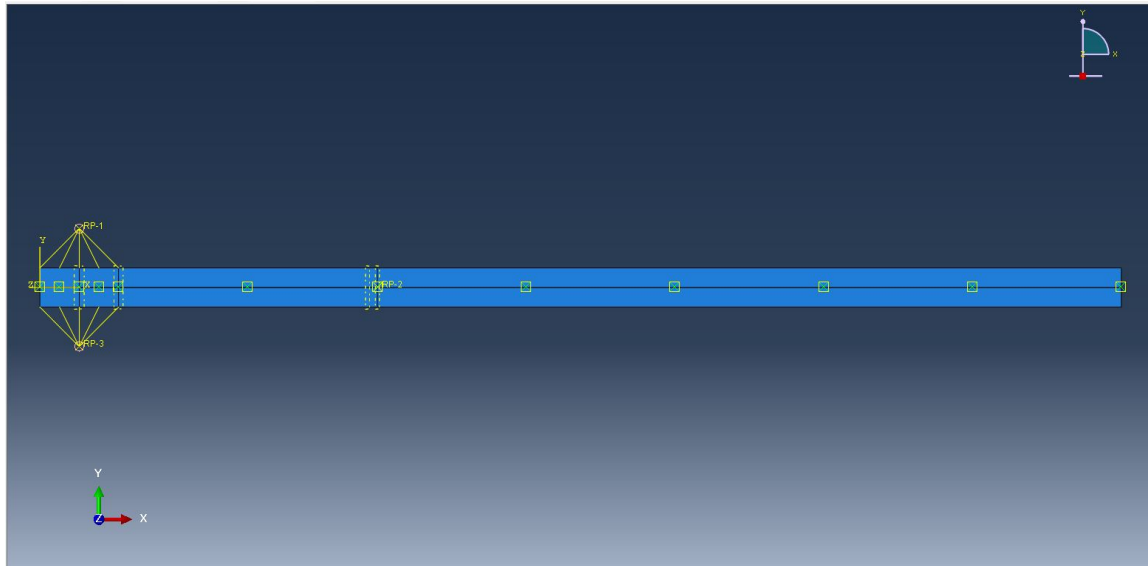
Interfacial fracture was simulated using the modelling strategies de-

scribed in Section 4.1, namely the Virtual Crack Closure Technique (VCCT) and the surface-based cohesive contact formulation.

The boundary conditions and interaction definitions adopted in the model are illustrated in Figure 4.16.



(a) Boundary conditions and loading configuration.



(b) Interfacial interaction definition.

Figure 4.16: Numerical setup of the DCB model: boundary conditions and interfacial interaction definitions.

For both interfacial modelling approaches, a uniform mesh size of 0.5 mm was adopted over the entire specimen, as shown in Figure 4.17. The selected mesh size represents a compromise between simulation accuracy and computational cost. Mesh sensitivity analyses showed that further refinement provided only limited improvements in peak prediction, while significantly increasing computational time.

A structured quadrilateral mesh was employed. The elements belong to the plane strain family and were defined as CPE4 elements (four-node bilinear plane strain quadrilateral elements), as visible in Figure 4.17.

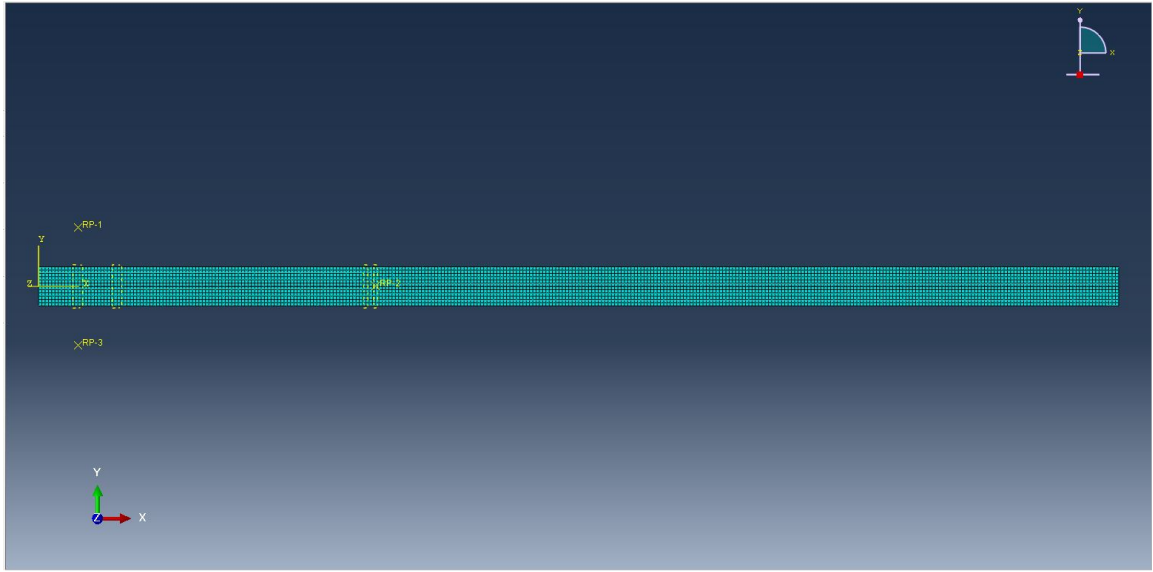


Figure 4.17: Finite element mesh of the DCB model.

4.4.2 Results and Comparison

Figure 4.18 shows the deformed configuration of the model. The vertical displacement along the y -direction is highlighted, clearly showing the symmetric deformation of the two arms during the opening phase.

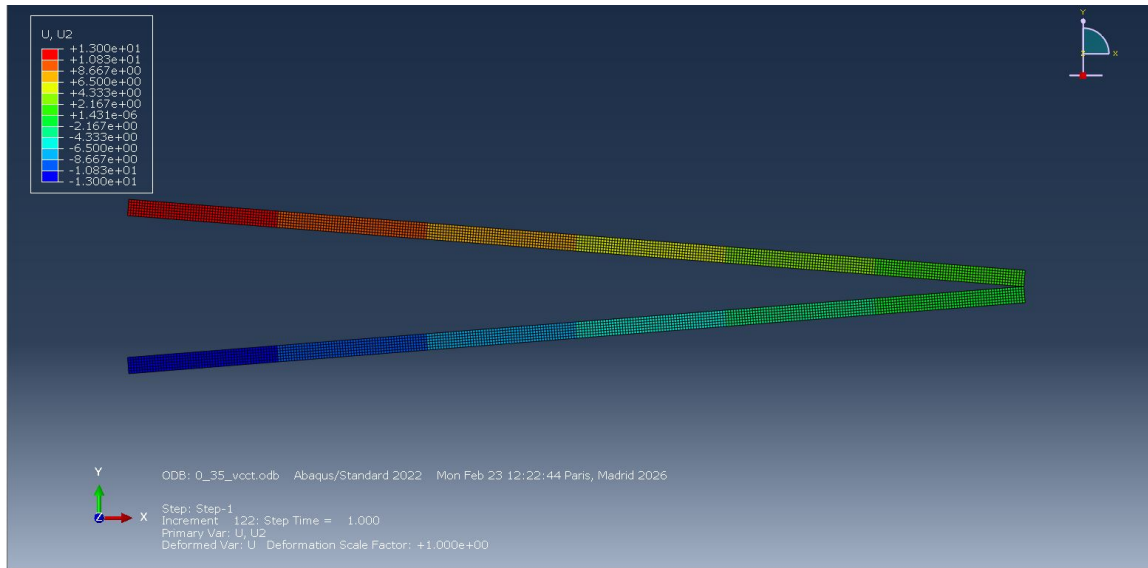


Figure 4.18: Deformed configuration of the DCB model.

The stress component σ_{22} (vertical normal stress) evaluated at the peak of the force–displacement curve is reported in Figure 4.19. It can be observed that the maximum tensile stress is concentrated at the crack tip. Following this stress concentration, the crack opens and propagation initiates along the predefined interface.

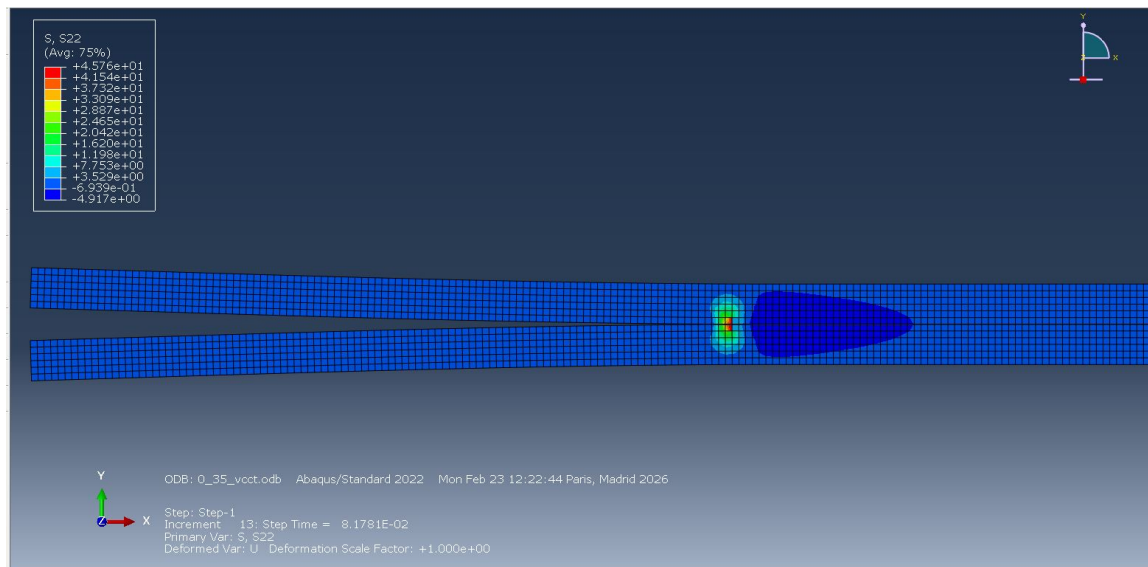


Figure 4.19: Distribution of σ_{22} at peak load.

In order to reproduce as faithfully as possible the behaviour observed

experimentally, the simulation was repeated three times by varying the value of the critical Mode I energy release rate G_{Ic} (0.25, 0.35 and 0.45 N/mm). This parametric study allowed the evaluation of the influence of fracture toughness on the global structural response and the identification of the value providing the best agreement with experimental force–displacement curves.

The comparison between numerical predictions (VCCT and surface-based cohesive contact) and experimental results is presented in Figures 4.20, 4.21 and 4.22. For each investigated value of G_{Ic} , both the complete force–displacement response and a zoom of the peak region are reported in order to better highlight the differences in terms of peak load prediction and post-peak behaviour.

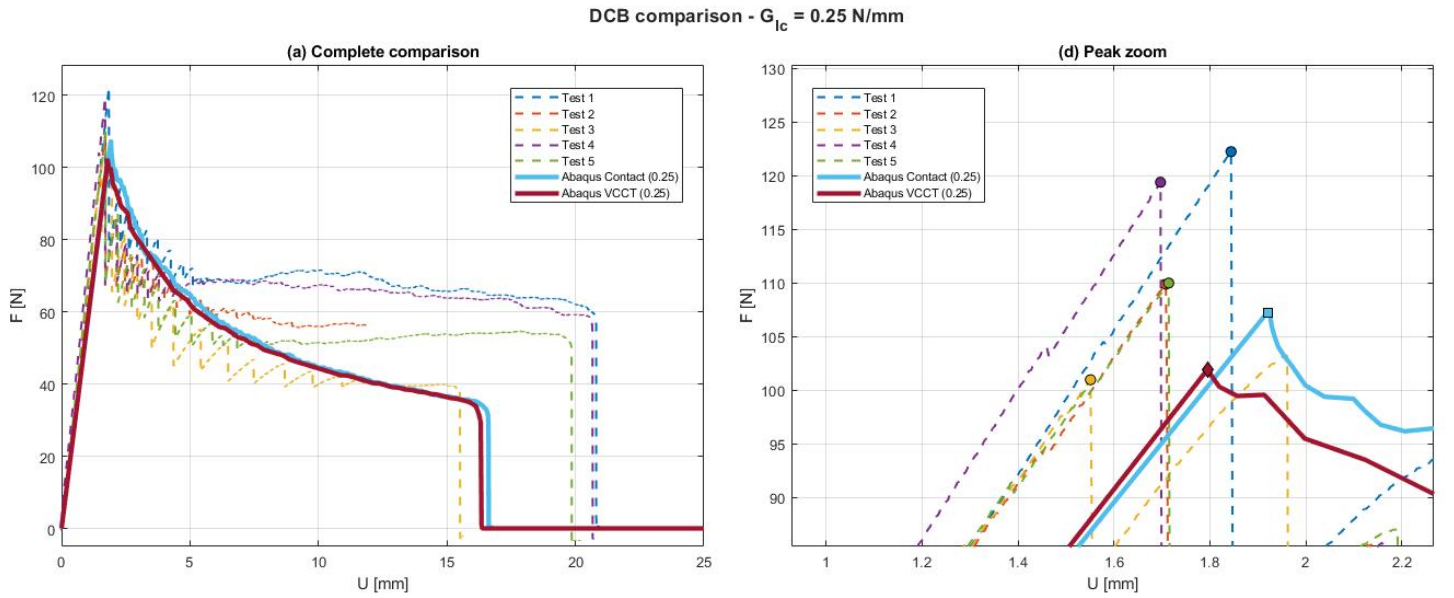


Figure 4.20: $G_{Ic} = 0.25$ N/mm: comparison between experimental and numerical force–displacement curves, including a zoom of the peak region.

DCB comparison - $G_{Ic} = 0.35 \text{ N/mm}$

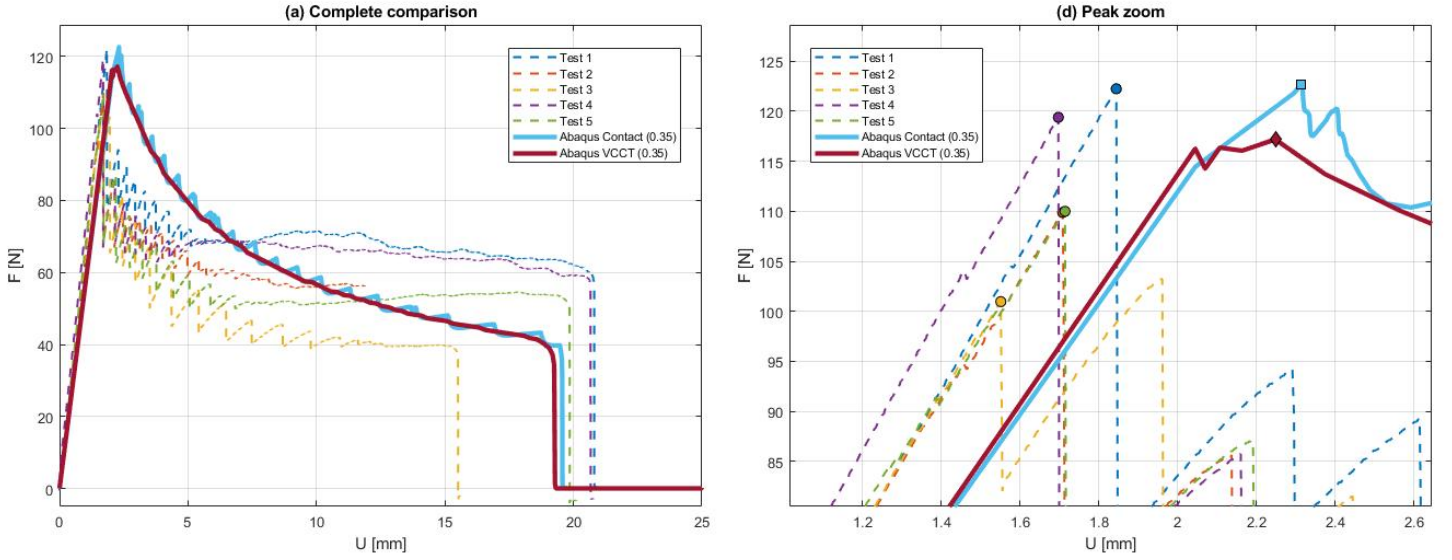


Figure 4.21: $G_{Ic} = 0.35 \text{ N/mm}$: comparison between experimental and numerical force-displacement curves, including a zoom of the peak region.

DCB comparison - $G_{Ic} = 0.45 \text{ N/mm}$

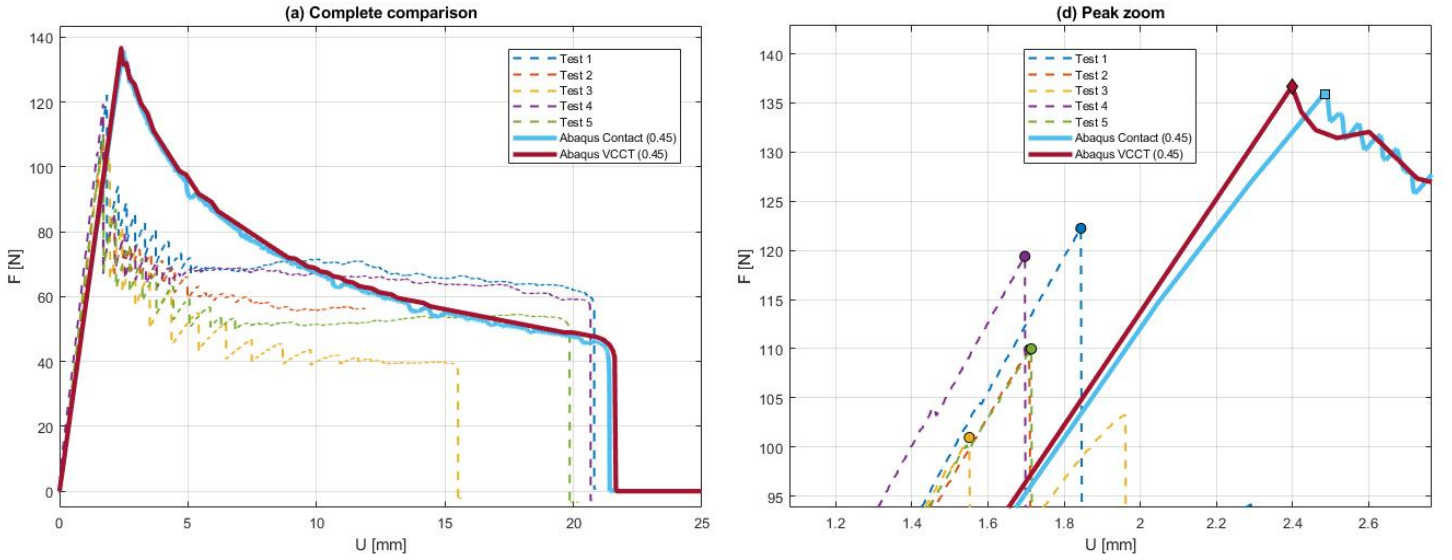


Figure 4.22: $G_{Ic} = 0.45 \text{ N/mm}$: comparison between experimental and numerical force-displacement curves, including a zoom of the peak region.

For $G_{Ic} = 0.25 \text{ N/mm}$ (Fig. 4.20), the peak force is underestimated with respect to some of the experimental results, particularly for Test 1 and

Test 4, as can be observed more clearly in the zoom of the peak region. Although the initial elastic phase remains qualitatively consistent with the experimental curves, the simulated stiffness appears slightly lower than the experimental one. This behaviour is somewhat unexpected, since a plane strain formulation would theoretically lead to a stiffer response. The discrepancy may be attributed to modelling assumptions, including boundary conditions, simplified two-dimensional representation, or uncertainties in the identification of elastic properties.

Nevertheless, the predicted peak load obtained with $G_{Ic} = 0.25$ N/mm remains close to the experimental values observed for Test 2, Test 3 and Test 5. In addition, the post-peak numerical response follows more closely the average trend of the experimental curves, remaining within the experimental envelope during most of the propagation phase.

In contrast, for $G_{Ic} = 0.45$ N/mm (Fig. 4.22), the peak force is overestimated, as highlighted in the zoomed view of the peak region. The higher fracture toughness delays crack propagation and results in an overly resistant structural response. The propagation phase occurs at larger displacements compared to the experimental curves, leading to an overall overprediction of the load-carrying capacity. Also in this case, the initial elastic stiffness remains slightly lower than the experimental response.

For the intermediate value $G_{Ic} = 0.35$ N/mm (Fig. 4.21), the predicted peak load is closer to the experimental values for Test 1 and Test 4, as visible in the peak zoom. However, for the remaining tests (Test 2, Test 3 and Test 5) the numerical peak appears slightly overestimated. Furthermore, during the post-peak phase the numerical curve tends to remain above the average experimental response, indicating a slightly overestimated resistance during crack propagation.

For all investigated values of G_{Ic} , the numerical simulations are unable to reproduce the initial unstable stepwise post-peak behaviour observed experimentally. The experimental curves exhibit a staircase-like response associated with unstable crack propagation, while the numerical model predicts a smoother and more regular crack growth. This difference can be attributed to the idealized nature of the numerical formulation, the absence of material heterogeneity, and the limitations of the two-dimensional modelling approach in capturing three-dimensional and instability effects present in the real specimens.

Overall, the parametric study highlights the strong influence of the fracture toughness parameter on both peak load and crack propagation behaviour. While $G_{Ic} = 0.35$ N/mm reproduces more accurately the peak load for Test 1 and Test 4, the value $G_{Ic} = 0.25$ N/mm provides a propagation behaviour that better follows the average experimental trend and remains more representative of the majority of the experimental curves. Furthermore, the value $G_{Ic} = 0.25$ N/mm is consistent with the fracture toughness identified from the experimental tests, reinforcing its interpretation as the

most representative effective fracture toughness for the adopted numerical model.

4.5 End-Notched Flexure (ENF)

4.5.1 Numerical Setup

In order to simulate the End-Notched Flexure test, a two-dimensional modelling approach was adopted by neglecting the specimen width and representing only its longitudinal section. This modelling strategy allows the evaluation of the structural response per unit width, while the out-of-plane dimension is considered when scaling numerical reaction forces for comparison with experimental data.

The numerical model consists of two deformable planar parts representing the upper and lower arms of the specimen, in order to define the interfacial region where delamination propagation occurs. Analytically rigid planar surfaces were used to model the lower support pins and the upper loading pin. Figure 4.23 shows a representative view of the numerical model geometry.

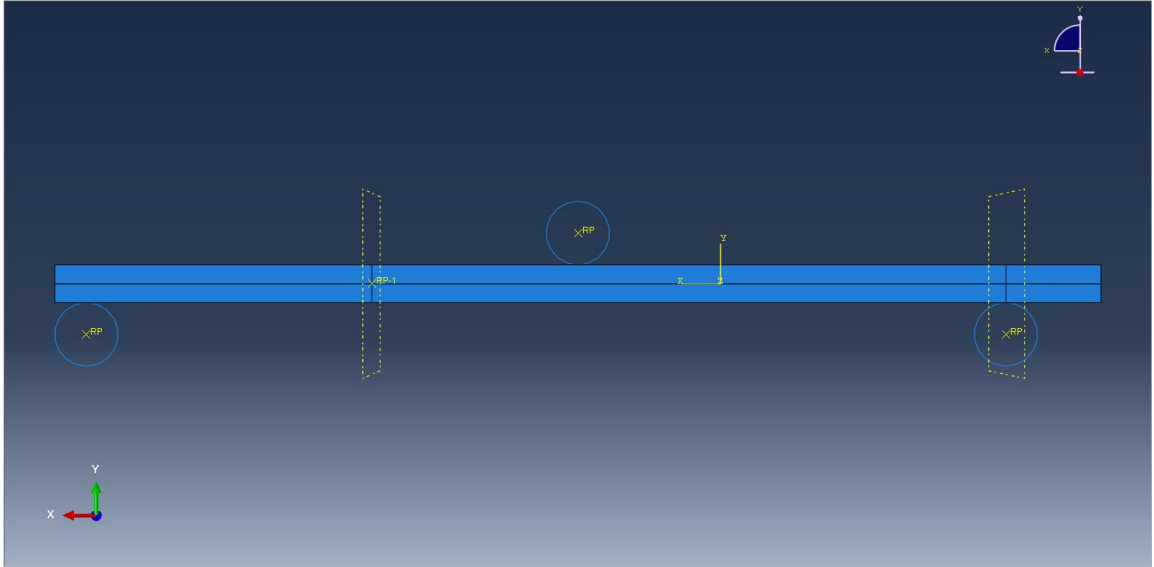


Figure 4.23: Finite element model of the ENF specimen.

The material properties are those reported in Table 4.1.

Interfacial fracture between the two arms was simulated using the modelling strategies described in Section 4.1, namely the Virtual Crack Closure Technique (VCCT) and the surface-based cohesive contact formulation.

Contact interactions were defined between the rigid pins and the specimen surfaces in order to prevent interpenetration during loading. The nor-

mal behaviour was defined as hard contact, while tangential behaviour was assumed frictionless, since friction effects were considered negligible for the present configuration.

In the initial step, the necessary translational degrees of freedom were constrained in order to prevent rigid body motion. In the subsequent loading step, the two lower support pins were fully constrained, while the upper loading pin was constrained in the horizontal direction and subjected to a prescribed vertical displacement in the negative y direction, reproducing the experimental bending configuration.

The boundary conditions adopted in the model are illustrated in Figure 4.24.

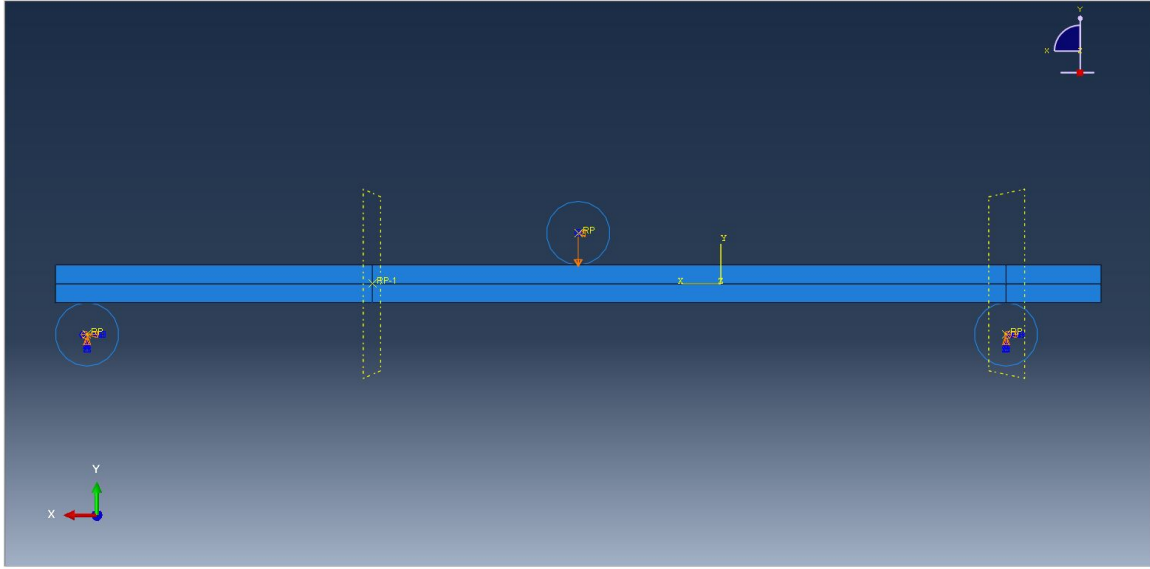


Figure 4.24: Boundary conditions and loading configuration of the ENF model.

For both interfacial modelling approaches, a uniform mesh size of 0.5 mm was adopted over the entire specimen, as shown in Figure 4.25. The selected mesh size represents a compromise between simulation accuracy and computational cost. Mesh sensitivity analyses showed that further refinement provided only limited improvements in peak prediction, while significantly increasing computational time.

A structured quadrilateral mesh was employed. The elements belong to the plane strain family and were defined as CPE4 elements (four-node bilinear plane strain quadrilateral elements), as visible in Figure 4.25.

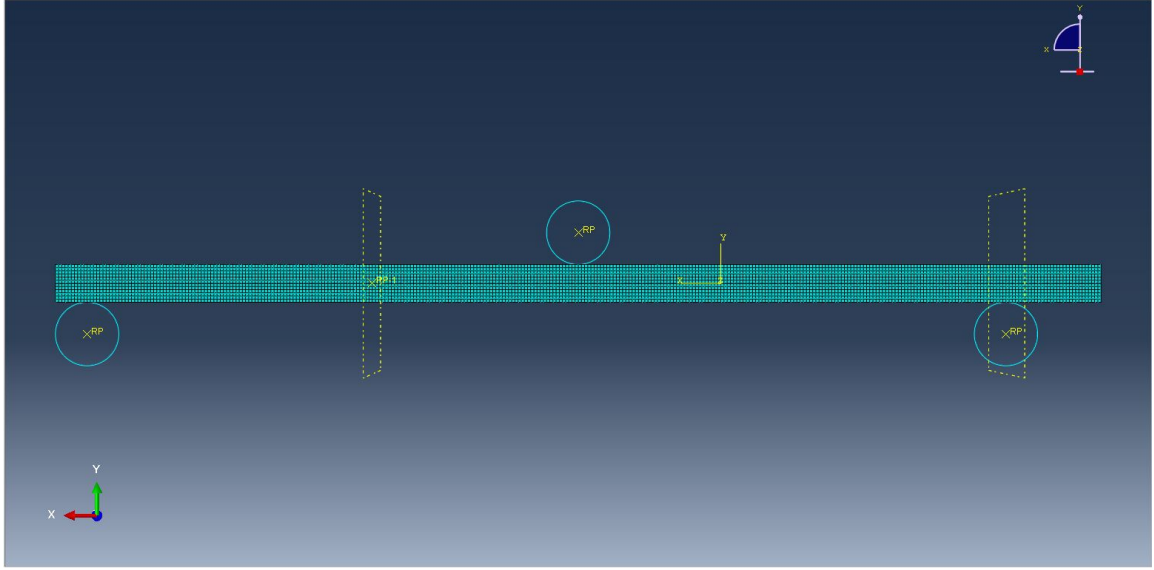


Figure 4.25: Finite element mesh of the ENF model.

4.5.2 Results and Comparison

Figure 4.26 shows the deformed configuration of the ENF model. The vertical displacement along the y -direction (U_2) is highlighted. The deformation is not symmetric, and the region where the crack is present exhibits a larger vertical displacement compared to the rest of the structure. This behaviour reflects the local reduction of stiffness introduced by the delaminated interface.

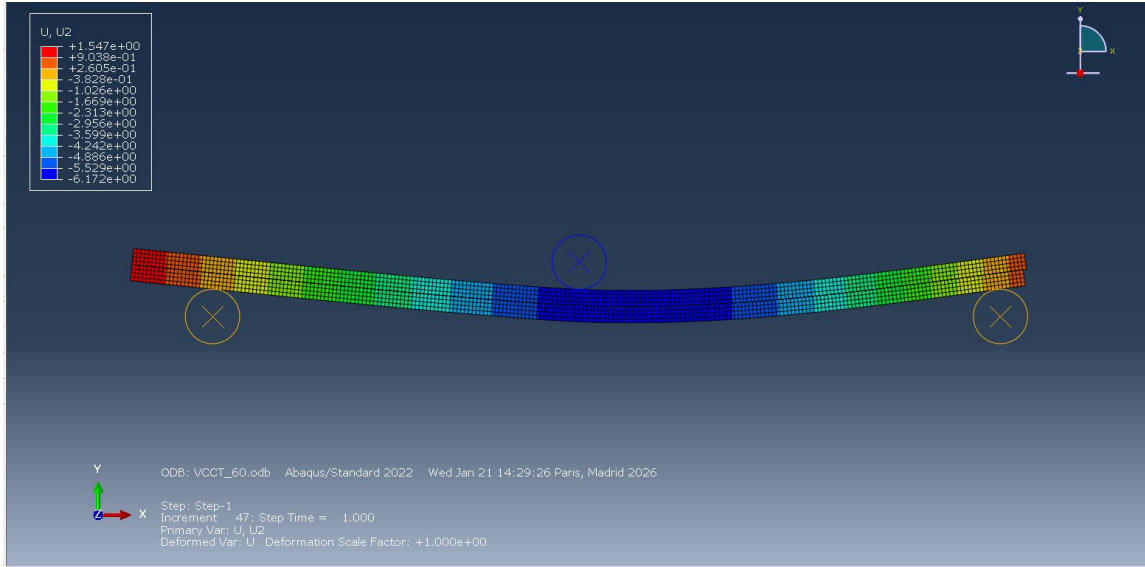


Figure 4.26: Deformed configuration of the ENF model (U2 displacement field).

The shear stress component σ_{12} evaluated at the peak of the force–displacement curve is reported in Figure 4.27. The shear stress increases sharply in the immediate vicinity of the crack tip, indicating a highly localized stress concentration. This localized stress field governs the initiation of delamination under predominant Mode II loading conditions.

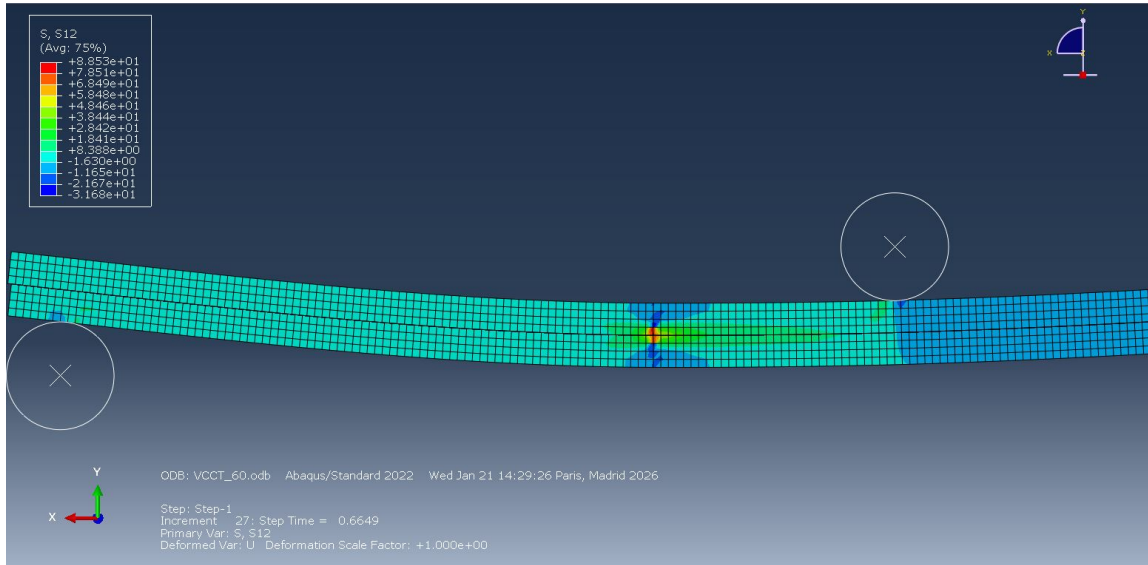


Figure 4.27: Distribution of σ_{12} at peak load for the ENF model.

The stress component σ_{22} (vertical normal stress) is reported in Figure 4.28. A pronounced compressive stress concentration can be observed beneath the central loading pin, generated by the contact pressure applied during the test. This region exhibits the highest compressive levels within the specimen and represents the dominant contact effect in the ENF configuration.

Although compressive stresses are also present beneath the support pins, their magnitude is significantly lower and therefore less influential on the global structural response. The numerical results are consistent with the experimental observations, where the loading pin introduces the primary localized compressive state in the specimen.

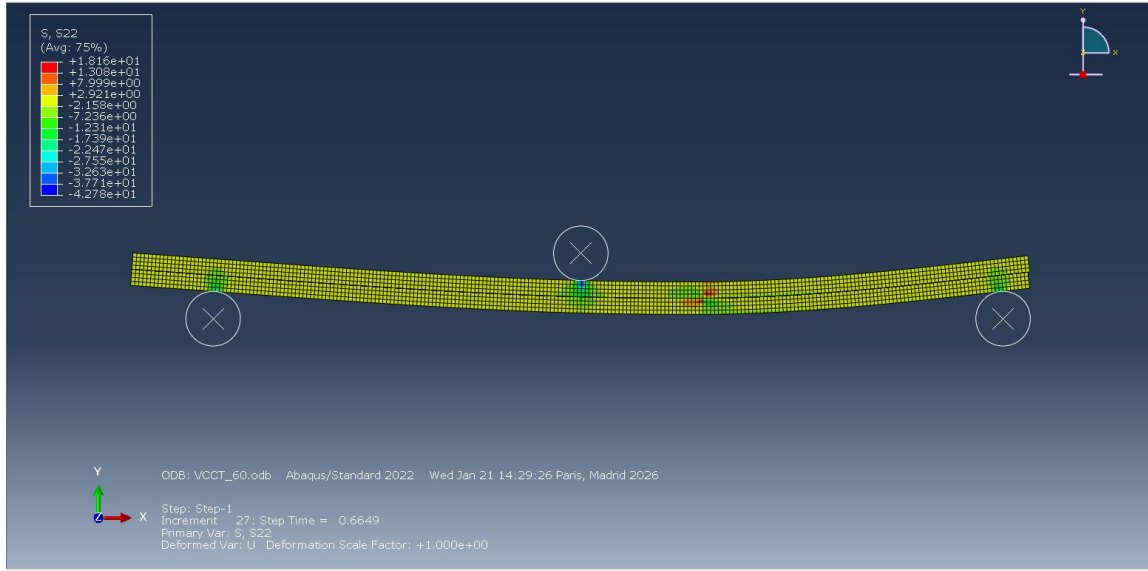
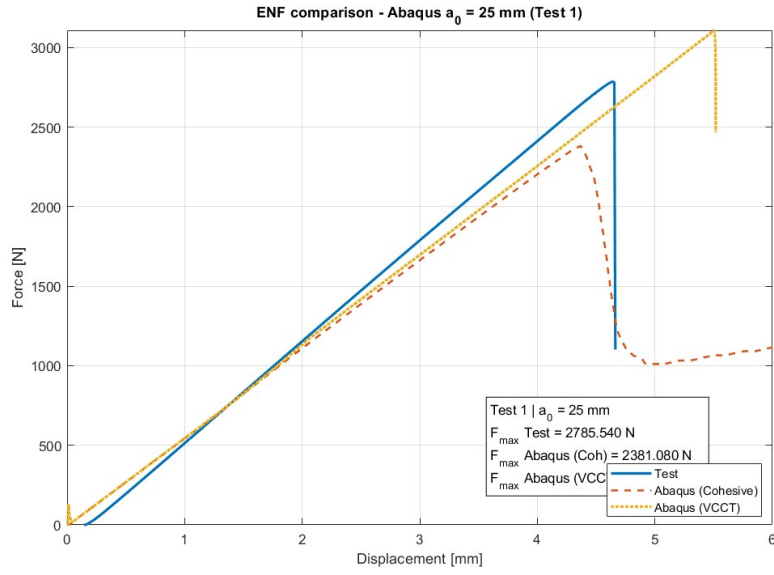
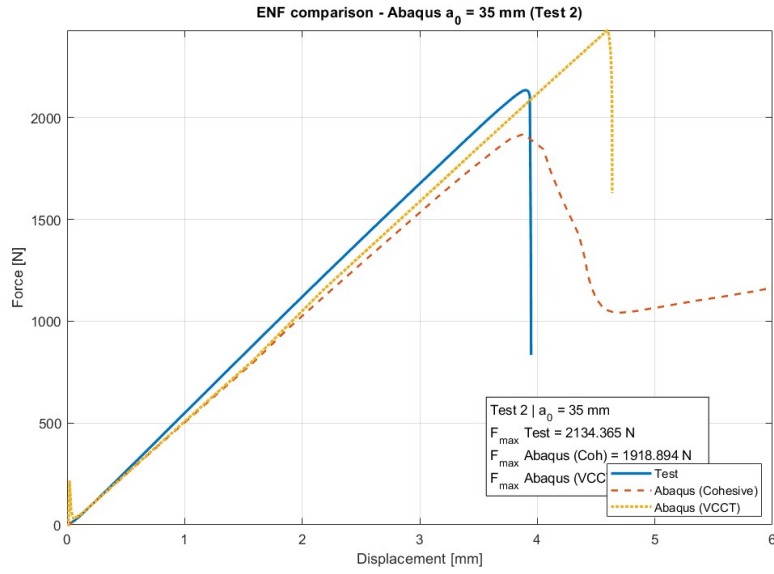


Figure 4.28: Distribution of σ_{22} in the ENF model.

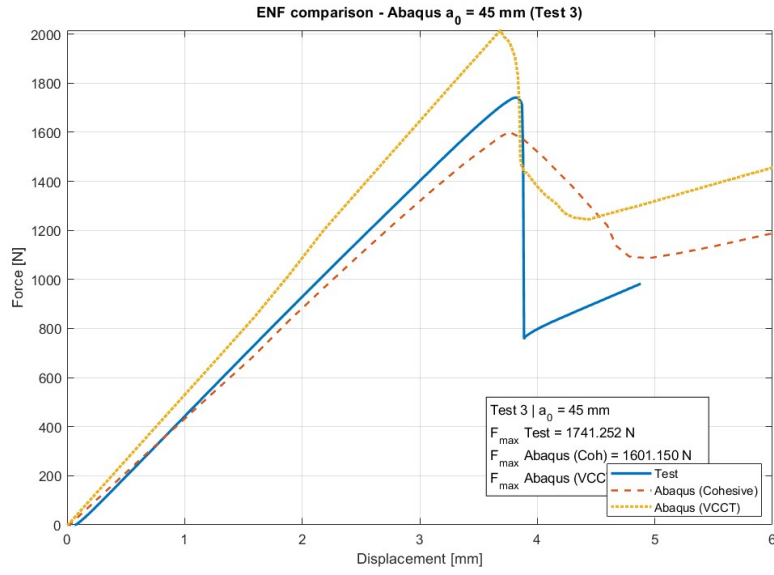


(a) $a_0 = 25$ mm – Test 1.

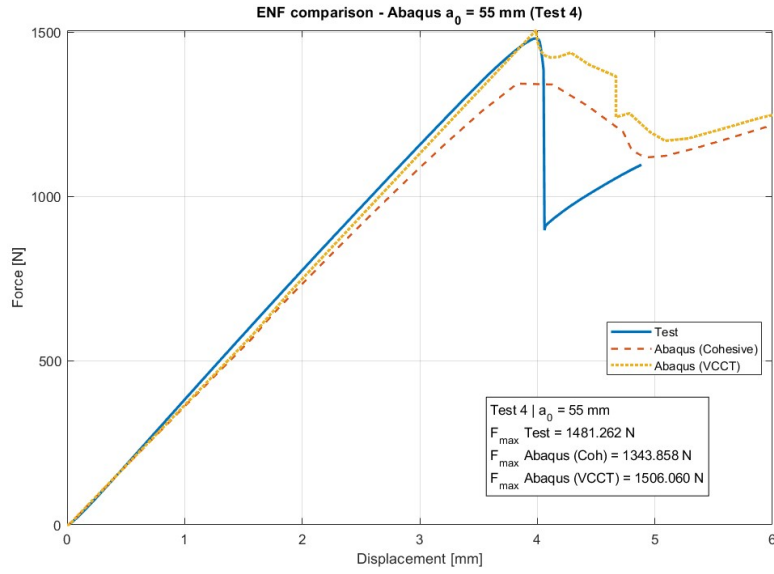


(b) $a_0 = 35$ mm – Test 2.

Figure 4.29: Comparison between experimental and numerical ENF force–displacement curves for different initial crack lengths a_0 (Tests 1 and 2).

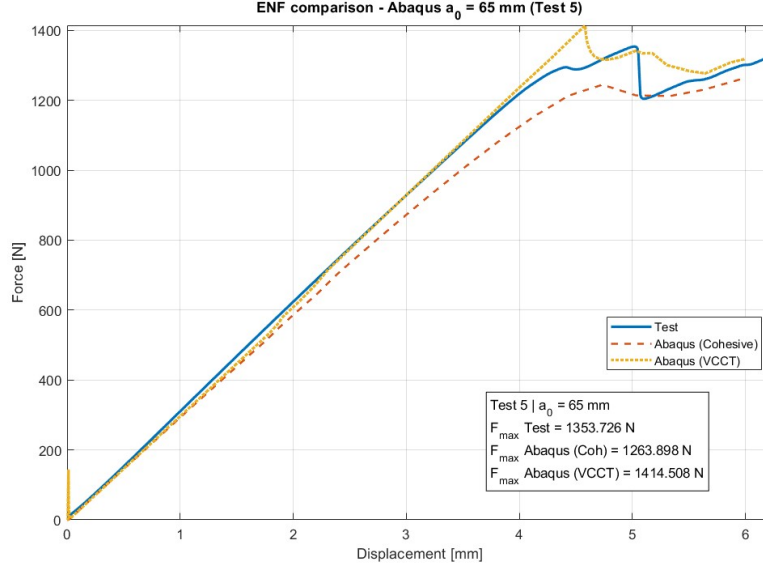


(a) $a_0 = 45$ mm – Test 3.



(b) $a_0 = 55$ mm – Test 4.

Figure 4.30: Comparison between experimental and numerical ENF force–displacement curves for different initial crack lengths a_0 (Tests 3 and 4).



(a) $a_0 = 65$ mm – Test 5.

Figure 4.31: Comparison between experimental and numerical ENF force–displacement curves for different initial crack lengths a_0 (Test 5).

Figures 4.29, 4.30, and 4.31 show the comparison between experimental and numerical force–displacement curves for different initial crack lengths a_0 . For each test, the corresponding experimentally identified value of G_{IIc} was adopted in the numerical model in order to reproduce as closely as possible the specific experimental configuration. Therefore, a different G_{IIc} value was used for each crack length. This choice does not imply a variation of the intrinsic material fracture toughness. Rather, the experimentally identified values are interpreted as effective, configuration-dependent parameters that incorporate the non-idealities of the ENF setup (e.g., instability, contact effects and local compressive stresses). The objective of the numerical campaign is therefore the validation of the structural response of each configuration, rather than the identification of a unique intrinsic G_{IIc} value.

In ENF configurations, the initial crack length plays a crucial role in the stability of delamination propagation. For the present setup, with a support span $L = 145$ mm, stable propagation is expected when

$$a_0 > 0.55 (L/2),$$

which corresponds to $a_0 \geq 40$ mm.

The investigated crack lengths range from 25 mm to 65 mm. The two shortest configurations ($a_0 = 25$ mm and $a_0 = 35$ mm) lie below the stability threshold, while the remaining cases satisfy the stability condition.

As the initial crack length increases from 25 mm to 65 mm, a progressive improvement in the agreement between numerical and experimental curves can be observed. For shorter cracks ($a_0 = 25$ mm and $a_0 = 35$ mm), the numerical models exhibit larger discrepancies in peak force prediction and in the displacement at crack initiation. In particular, the VCCT formulation shows greater difficulty in accurately reproducing the behaviour of shorter cracks, which are associated with more pronounced instability during crack propagation.

For larger crack lengths ($a_0 \geq 45$ mm), the global response becomes less sensitive to instability effects and local stress concentrations. Both modelling approaches show improved convergence towards the experimental curves. In these cases, the VCCT method generally provides a more accurate prediction of the peak load and better captures the onset of instability.

Regarding structural stiffness, the numerical curves remain reasonably consistent with the experimental slopes, although minor discrepancies persist. These differences may be attributed to modelling simplifications, the two-dimensional plane strain representation of the specimen, and the idealization of contact conditions.

In all cases, the post-peak response is not fully reproduced. The experimental curves exhibit a more abrupt instability after the peak load, whereas the numerical simulations tend to provide a smoother and more stabilized response. This behaviour may be partially influenced by the numerical stabilization techniques adopted in the model, such as the introduction of viscosity, which can dampen the abruptness of crack propagation.

Overall, increasing the initial crack length results in a more reproducible numerical response. The VCCT formulation globally provides a closer prediction of peak force and instability behaviour for larger crack lengths, while the cohesive formulation tends to produce a more regular and stabilized post-peak evolution compared to the experimental response.

4.6 Four-Point End-Notched Flexure (4ENF)

4.6.1 Numerical Setup

In order to simulate the Four-Point End-Notched Flexure test, a two-dimensional modelling approach was adopted by neglecting the specimen width and representing only its longitudinal section. This modelling strategy allows the evaluation of the structural response per unit width, while the out-of-plane dimension is considered when scaling numerical reaction forces for comparison with experimental data.

The numerical model consists of two deformable planar parts representing the upper and lower arms of the specimen, in order to define the interfacial region where delamination propagation occurs. Analytically rigid planar surfaces were used to model the lower support pins and the two up-

per loading pins. Figure 4.32 shows a representative view of the numerical model geometry.

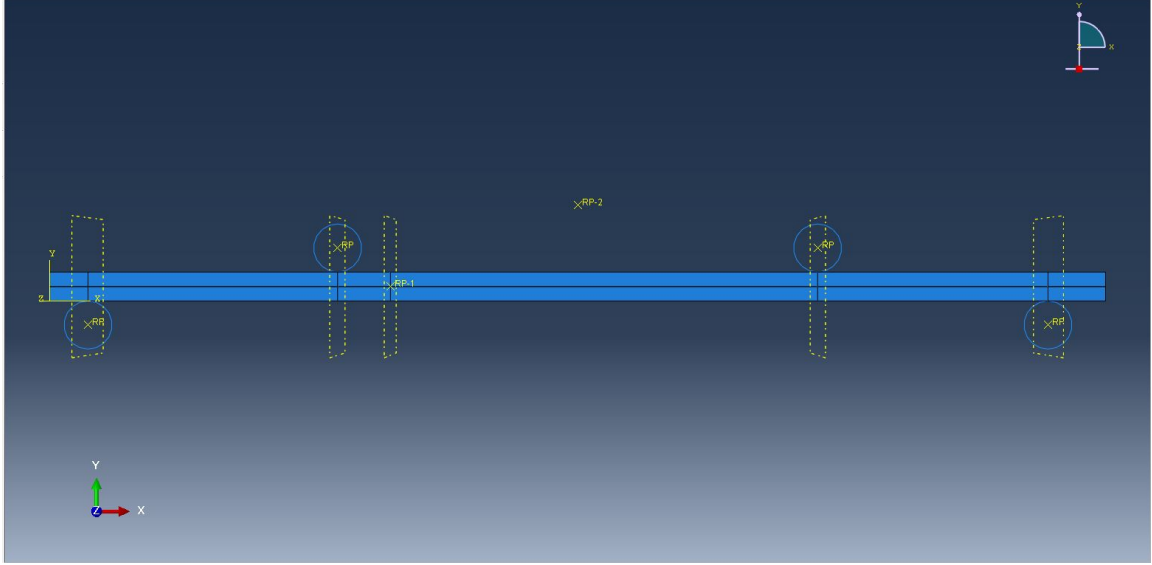


Figure 4.32: Finite element model of the 4ENF specimen.

The material properties are those reported in Table 4.1.

Interfacial fracture between the two arms was simulated using the modelling strategies described in Section 4.1, namely the Virtual Crack Closure Technique (VCCT) and the surface-based cohesive contact formulation.

Contact interactions were defined between the rigid pins and the specimen surfaces in order to prevent interpenetration during loading. The normal behaviour was defined as hard contact, while tangential behaviour was assumed frictionless, since friction effects were considered negligible for the present configuration.

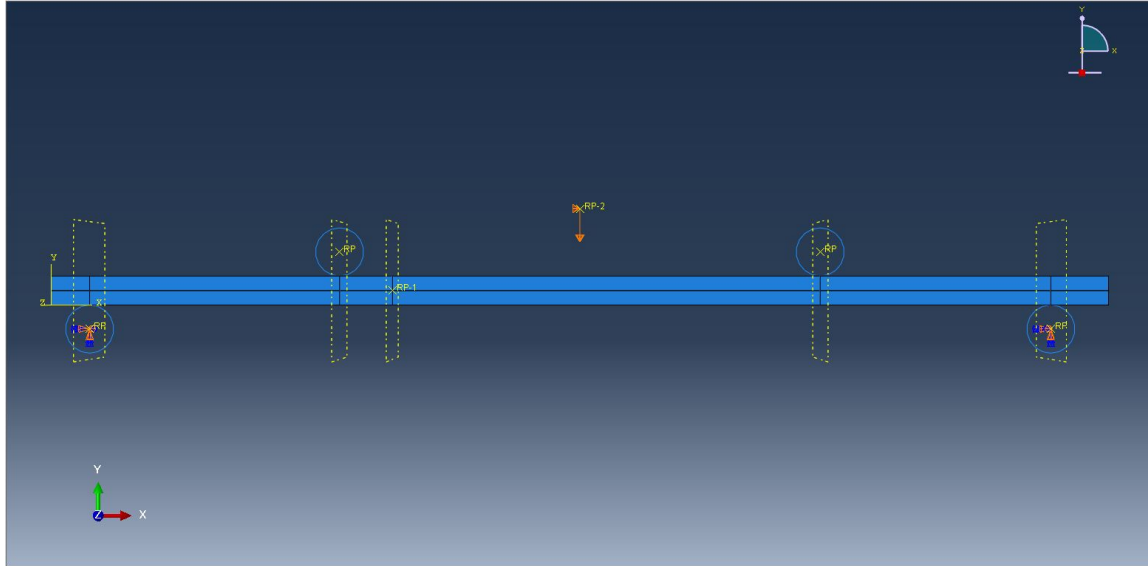
A reference point was created at the midspan between the two upper loading pins. A kinematic coupling constraint was defined between this reference point and the two upper pins. The prescribed vertical displacement was applied at the reference point, ensuring symmetric load transfer to both loading pins.

The two lower support pins were fully constrained. The two upper loading pins were constrained in the horizontal direction, while their rotation about the out-of-plane axis was left free. This rotational freedom allows the pins to accommodate the differential deformation of the specimen near the crack region, preventing artificial loss of contact due to local slope variations.

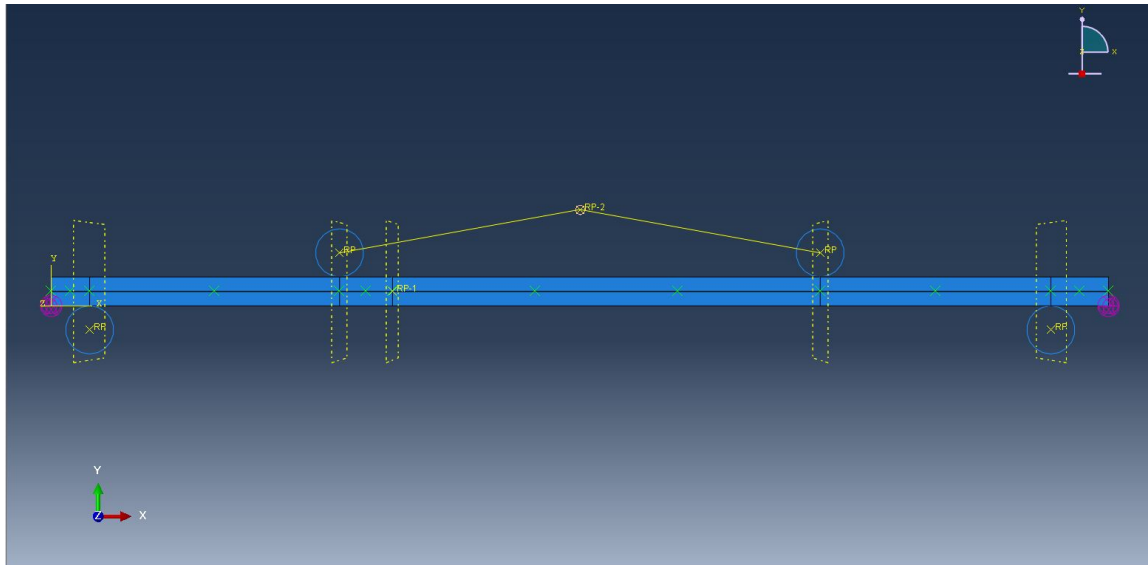
In the initial step, the necessary translational degrees of freedom were constrained in order to prevent rigid body motion. In the subsequent loading step, a prescribed vertical displacement in the negative y direction was ap-

plied at the reference point, reproducing the experimental four-point bending configuration.

The adopted boundary conditions and loading configuration are illustrated in Figure 4.33.



(a) Boundary conditions and loading configuration.



(b) Interfacial interaction definition.

Figure 4.33: Numerical setup of the 4ENF model: boundary conditions and interfacial interaction definitions.

For both interfacial modelling approaches, a uniform mesh size of 0.5 mm was adopted over the entire specimen, as shown in Figure 4.34. The selected mesh size represents a compromise between simulation accuracy and computational cost. Mesh sensitivity analyses showed that further refinement provided only limited improvements in peak prediction, while significantly increasing computational time.

A structured quadrilateral mesh was employed. The elements belong to the plane strain family and were defined as CPE4 elements (four-node bilinear plane strain quadrilateral elements), as visible in Figure 4.34.

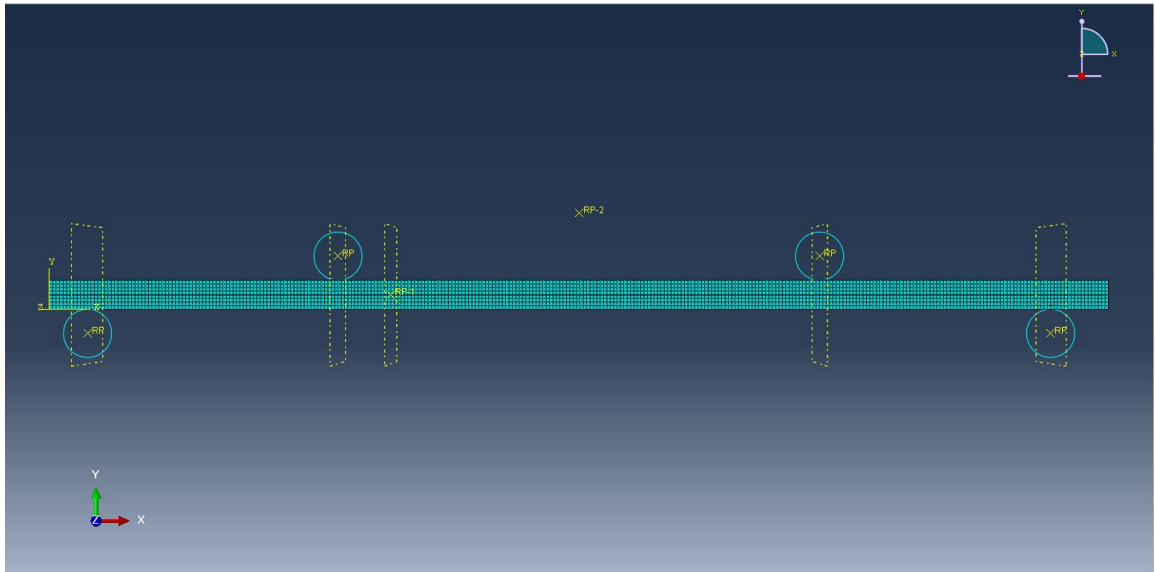


Figure 4.34: Finite element mesh of the 4ENF model.

4.6.2 Results and Comparison

Figure 4.35 shows the deformed configuration of the 4ENF specimen under the applied load of the two upper pins. The deformation pattern is not uniform along the beam length. Similarly to the classical ENF configuration, a larger vertical displacement is observed in the region where the initial crack is located. This behaviour is associated with the reduced bending stiffness of the structure in the cracked region, which locally increases the compliance of the specimen.

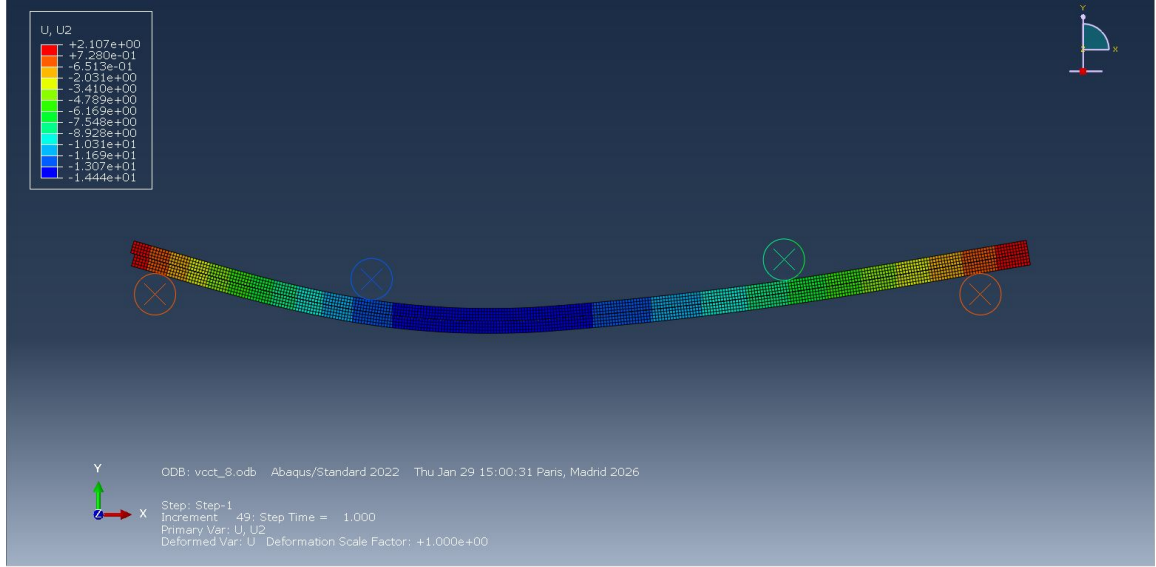


Figure 4.35: Deformed configuration of the 4ENF specimen under loading (U_2 displacement field).

The shear stress distribution S_{12} at peak load is reported in Figure 4.36. As expected for a Mode II dominated configuration, the shear stress is strongly concentrated at the crack tip. A localized compressive region can be observed at the crack apex, surrounded above and below by regions of maximum tensile shear stress.

This sign change in S_{12} is the combined effect of interfacial sliding and local bending rotation of the specimen arms near the crack front. The stress concentration remains highly localized in the crack tip region, while away from the crack the shear stress distribution appears relatively uniform along the central portion of the beam. This shear component represents the driving mechanism for crack initiation and subsequent delamination propagation in the 4ENF test.

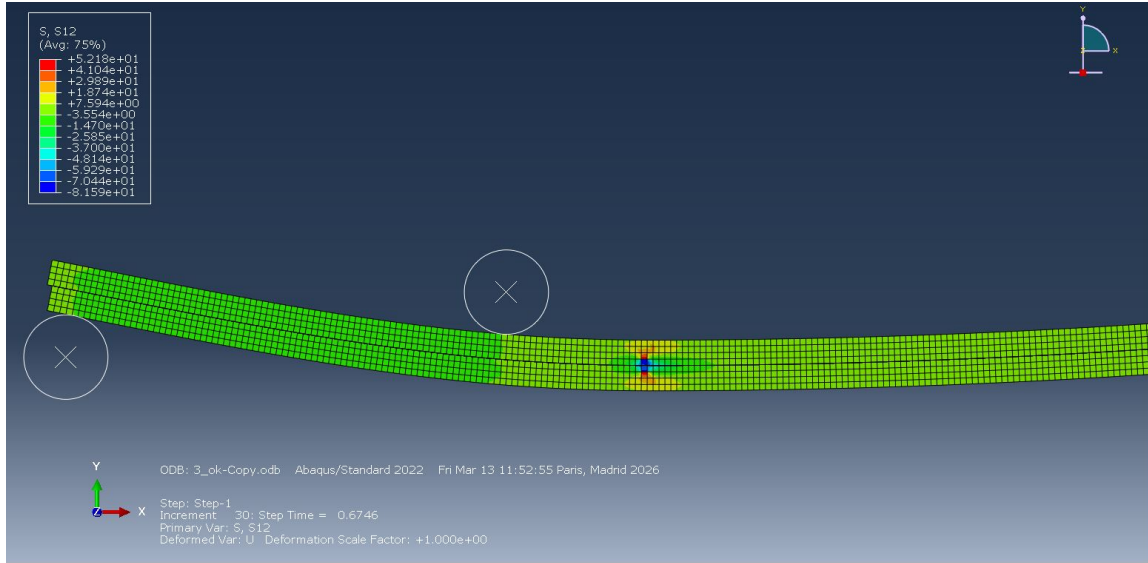


Figure 4.36: Shear stress distribution σ_{12} at peak load in the 4ENF specimen.

Figure 4.37 shows the transverse normal stress component S_{22} . A clear local compressive state is observed beneath the loading pins, induced by the contact pressure transmitted by the rigid indenter surfaces. This compressive stress is superimposed on the global bending response of the beam. In the vicinity of the crack tip, a localized tensile region can also be observed, resulting from the local stress redistribution associated with crack tip rotation and shear-dominated deformation.

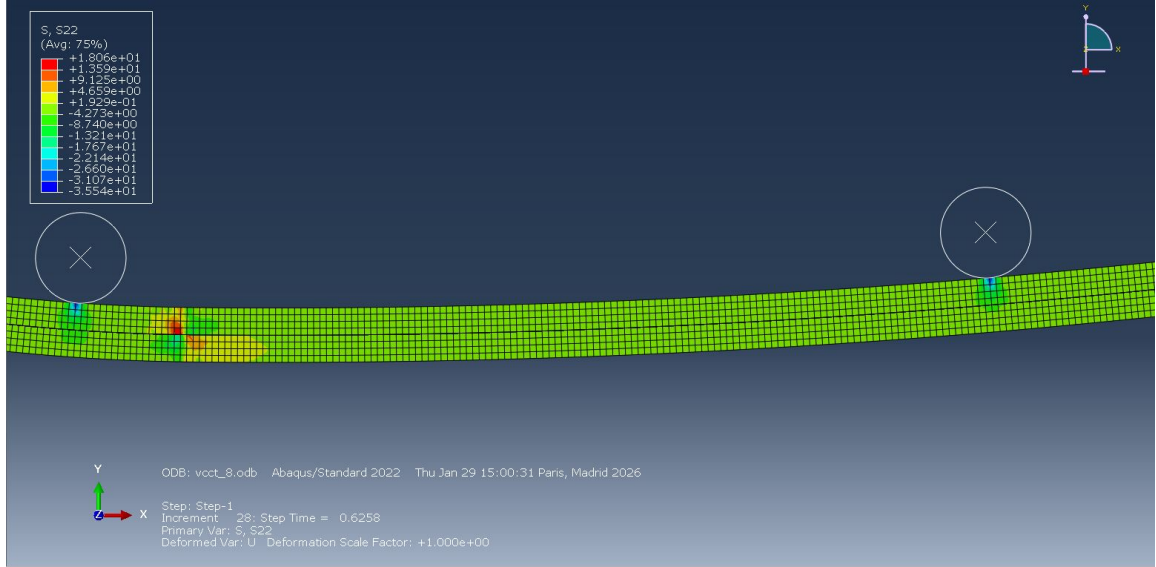


Figure 4.37: Transverse normal stress distribution σ_{22} in the 4ENF specimen.

The comparison between the experimental and numerical force–displacement curves is presented in Figures 4.38 and 4.39. In the numerical simulations, the interlaminar fracture toughness values identified experimentally were directly implemented for each test. In particular, a value of $G_{IIC} = 3.4$ was used for Test 1, while $G_{IIC} = 3.7$ was adopted for Tests 2 and 3.

As observed in the figures, the peak force is reproduced with good agreement by both numerical approaches (VCCT and cohesive contact). However, discrepancies can be noticed in the displacement corresponding to the peak load. The numerical curves exhibit a slightly higher displacement at peak, indicating a lower global stiffness compared to the experimental response.

Such differences can be attributed to the inherent idealizations introduced in the numerical modelling as well as to the unavoidable variability associated with experimental measurements. Despite these deviations, the overall trend of the response and the magnitude of the peak load are consistently captured.

Moreover, the numerical results show a more extended stability plateau during crack propagation in the 4ENF configuration. In the experimental curves, this stable propagation region appears significantly shorter. This behaviour is consistent with the idealized nature of the numerical model, which does not account for all the complexities that may influence crack evolution in real specimens.

Overall, the numerical simulations provide a coherent representation of the experimental behaviour, successfully reproducing the main mechanical features of the 4ENF test.

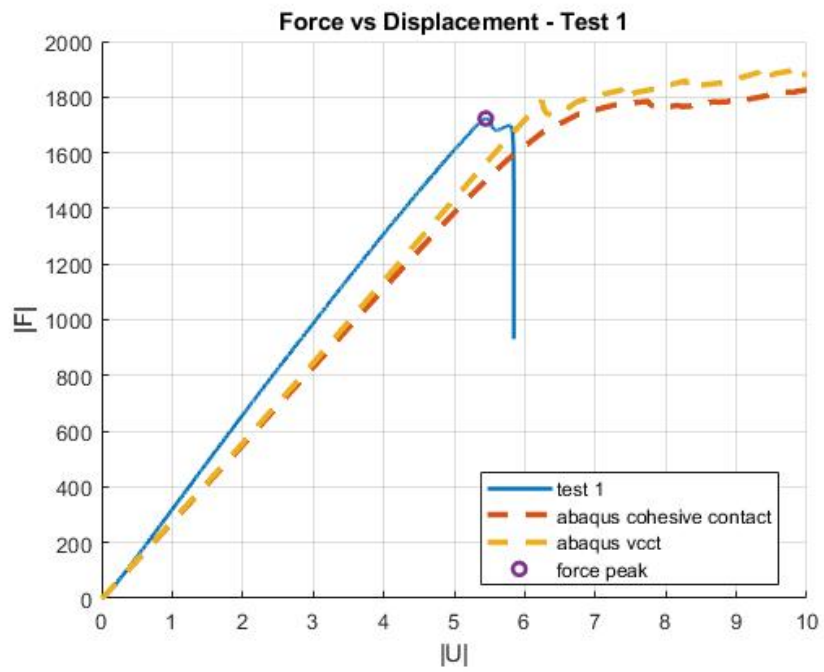


Figure 4.38: Comparison between experimental and numerical force–displacement curves for Test 1.

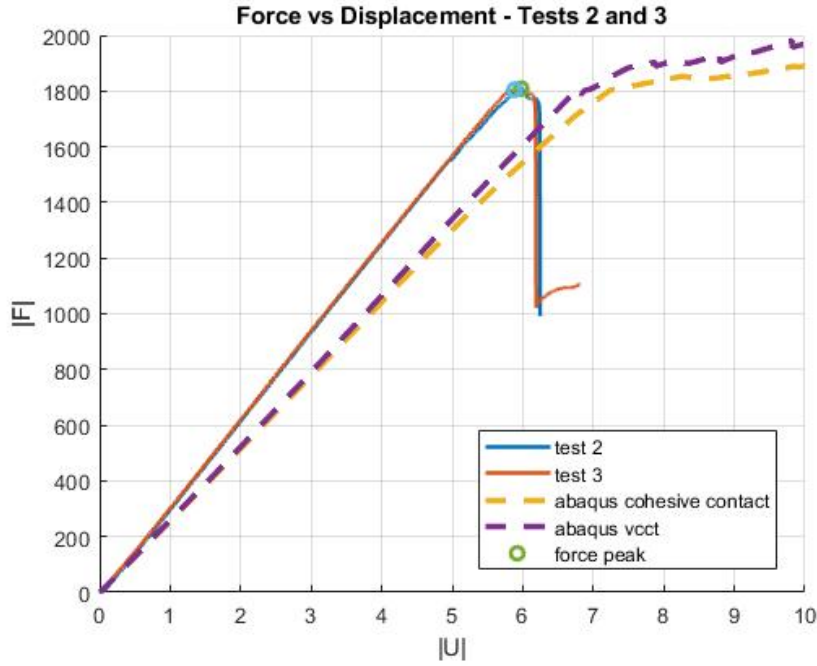


Figure 4.39: Comparison between experimental and numerical force–displacement curves for Tests 2 and 3.

4.7 Discussion of Numerical Results

In this chapter, the results obtained from the numerical simulations developed in Abaqus have been presented, with the aim of reproducing the mechanical response of the specimens observed during the experimental tests and evaluating the capability of the numerical models to represent the experimental behaviour.

With regard to the models developed for the tensile test and the three-point bending (3PB) test, the analysis was limited to the elastic response only, without introducing any damage or failure criteria. In the case of the tensile test, the elastic behaviour was reproduced satisfactorily, confirming that the definition of the laminate layup represents an effective approach for modelling the mechanical behaviour of composite laminates. For the 3PB test, the global structural response was also reasonably reproduced, although some discrepancies were observed in the stiffness of the force–displacement curve.

For the tests aimed at investigating interlaminar fracture (DCB, ENF and 4ENF), two different numerical approaches were adopted: the Virtual Crack Closure Technique (VCCT) and a surface-based cohesive contact formulation. In all the analysed cases, both numerical strategies were able to

reproduce satisfactorily the general trend of the force–displacement curves, particularly the maximum load associated with the onset of delamination propagation.

However, neither of the two methods was able to fully reproduce the unstable behaviour observed experimentally during delamination propagation. In the case of the DCB test, the numerical model does not reproduce the energy drops observed immediately after the initiation of crack propagation, showing instead a smoother and more regular crack growth. For the ENF tests, the VCCT method showed some difficulties in representing the tests characterised by shorter initial cracks, which experimentally correspond to more unstable responses. The cohesive contact approach generally provided a good representation of the elastic response of the specimen, although it tends to regularise the unstable part of the response.

For the 4ENF test, the numerical simulations showed a slightly lower stiffness compared to the experimental results. The VCCT method provided a slightly better prediction of the elastic response, although it still underestimated the overall stiffness of the system. After the onset load, both numerical approaches predicted a more stable crack propagation compared to the experimental observations, leading to a longer plateau region than the one observed in the experimental curves.

Overall, despite some local discrepancies, the numerical models developed in this work were able to reproduce satisfactorily the global response of the specimens, particularly in the elastic regime. Based on the results obtained, it is not possible to identify a single numerical method as clearly superior for the representation of these phenomena. Nevertheless, it can be observed that the VCCT approach tends to show greater difficulties in tests characterised by higher instability and may lead to convergence issues. On the other hand, VCCT is generally computationally less expensive and requires a smaller number of parameters for model definition.

The cohesive contact approach, instead, proved to be more robust from a numerical point of view and more stable in terms of convergence, although at the cost of a higher computational effort and the need to define a larger number of parameters to characterise the cohesive behaviour.

Chapter 5

High Strain Rate Behaviour of Composite Materials

5.1 Introduction

Composite materials used in aerospace structures are subjected to a wide range of loading conditions during their service life. In addition to conventional quasi-static loads, these structures may experience dynamic loading scenarios associated with impact events [18]. Typical examples include bird strikes on aircraft structures or debris impacts on engine blades, which can generate very high loading rates [19].

For this reason, it is essential to investigate the mechanical behaviour of composite materials when subjected to high strain rate loading conditions.

Several studies available in the literature have analysed the influence of strain rate on the mechanical properties of carbon fibre reinforced polymers (CFRP). It has been shown that unidirectional CFRP laminates loaded in longitudinal tension generally exhibit a relatively low strain rate sensitivity. This behaviour is mainly attributed to carbon fibers, which dominate the load carrying mechanism and are known to be weakly sensitive to strain rate effects [20].

However, other investigations have demonstrated that the strain rate sensitivity of composite laminates can increase when the fibre orientation and the stacking sequence are modified. In particular, when fibres are oriented away from the loading direction (e.g. 45° or 90°), the mechanical response becomes increasingly governed by the polymer matrix rather than the fibres. Since the matrix exhibits a viscoelastic behaviour, the laminate response may become significantly more sensitive to strain rate effects [21]. Consequently, the strain rate sensitivity of CFRP laminates strongly depends on the fibre orientation and on the laminate stacking sequence.

Furthermore, the mechanical response of unidirectional CFRP laminates may also differ depending on the type of loading applied. In particular, ex-

perimental results reported in the literature show that the behaviour under compressive loading can differ from that observed in tensile loading conditions [20].

In this chapter, a brief overview of the behaviour of composite materials subjected to high strain rate loading conditions is presented, with particular emphasis on CFRP laminates. The operating principle of the Split Hopkinson Pressure Bar (SHPB), which represents one of the most widely used experimental techniques for dynamic material characterisation, is also introduced.

5.2 Strain Rate Effects in CFRP Laminates

This section analyses the strain rate sensitivity of carbon fibre reinforced polymer (CFRP) laminates based on results reported in the literature. Particular attention is given to the variation of Young's modulus, tensile strength and failure mechanisms for different laminate configurations, considering both fibre orientation and stacking sequence.

5.2.1 Longitudinal Tensile Behaviour of Unidirectional Laminates (0°)

Unidirectional CFRP laminates with fibres oriented along the loading direction (0°) represent a fibre-dominated configuration. In this case, the mechanical response is primarily controlled by the carbon fibres, while the polymer matrix mainly contributes to load transfer between fibres.

Experimental studies performed within a strain rate range between 10^{-3} and 100 s^{-1} show that the stress-strain response remains almost linear elastic until failure for quasi-static loading conditions (10^{-3} and 10^{-2} s^{-1}). At intermediate strain rates (around 1 s^{-1}), small oscillations may appear in the stress-strain curves, which are generally attributed to vibrations in the testing equipment.

For the highest strain rates (10 and 100 s^{-1}), some experimental difficulties have been reported, such as specimen slippage from the gripping region, which may invalidate certain tests. In many cases, failure occurred close to the clamping region, where stress concentrations are typically present. The observed failure mechanism corresponds to fibre fracture, which tends to occur in multiple fragments as the strain rate increases.

Overall, only limited variations in tensile strength and Young's modulus were observed with increasing strain rate. This behaviour confirms that the mechanical response of unidirectional laminates loaded along the fibre direction is mainly governed by the carbon fibres, which exhibit a relatively low strain rate sensitivity [21].

5.2.2 Off-Axis Tensile Behaviour (45° and 90°)

When the fibre orientation deviates from the loading direction, such as in 45° and 90° laminates, the deformation mechanisms progressively involve the polymer matrix and the fibre–matrix interface.

Experimental results show a significant increase in tensile strength with increasing strain rate for both configurations. In particular, an increase of approximately 36% was observed for 45° laminates, while an increase of approximately 114% was reported for 90° laminates.

In contrast, the Young’s modulus showed only minor variations with strain rate. As expected, the tensile strength values for 45° and especially for 90° laminates remain significantly lower than those observed in the 0° configuration, since the fibres are not aligned with the loading direction.

The failure mechanisms observed in these laminates differ from the fibre fracture typical of the 0° configuration. Instead, damage mechanisms such as matrix cracking and fibre–matrix interfacial debonding are commonly observed. These failure modes usually occur within the gauge section of the specimen and are observed across the different strain rate levels. These observations confirm that the strain rate sensitivity observed in off-axis laminates is mainly associated with the viscoelastic behaviour of the polymer matrix [21].

5.2.3 Cross-Ply Laminates

Cross-ply laminates with stacking sequences $[0/90]_{2s}$ and $[-45/45]_{2s}$ were also investigated under the same strain rate range considered in the previous configurations.

For the $[0/90]_{2s}$ laminates, the stress–strain response remains predominantly linear elastic until failure, which typically occurs in a brittle manner. As the strain rate increases, oscillations may appear in the stress–strain curves due to vibrations in the testing equipment.

In contrast, the $[-45/45]_{2s}$ laminates exhibit a more complex mechanical response. After an initial linear elastic region, a pseudo-plastic behaviour can be observed. In this case, the laminate does not immediately lose its load carrying capacity after the onset of damage. Instead, the stress continues to increase progressively.

This behaviour is associated with progressive matrix damage and partial fibre–matrix debonding. During the test, some fibres lose their constraint with the surrounding matrix and may reorient under the applied shear stresses, allowing the laminate to continue sustaining load.

A comparison between the two configurations highlights a clear difference in terms of strength and failure strain. The $[0/90]_{2s}$ laminates exhibit higher tensile strength but fail at lower strain due to the presence of plies oriented along the loading direction, which significantly contribute to the

load-carrying capacity of the laminate. In contrast, the $[-45/45]_{2s}$ laminates show lower tensile strength but significantly larger failure strains. This behaviour is mainly related to the progressive degradation of the matrix and the shear-dominated deformation mechanisms that occur in the $\pm 45^\circ$ plies, allowing a more gradual damage evolution.

When analysing the influence of strain rate on the mechanical properties, the Young's modulus remains almost unaffected by the loading rate for both laminate configurations. In contrast, the tensile strength shows a slight increasing trend with strain rate for the $[0/90]_{2s}$ laminates, while remaining approximately constant for the $[-45/45]_{2s}$ laminates. These results highlight the strong influence of fibre orientation and stacking sequence on the strain rate sensitivity of CFRP laminates [21].

5.2.4 Longitudinal Compressive Behaviour of Unidirectional Laminates (0°)

Unidirectional CFRP laminates loaded in compression along the fibre direction (0°) represent a fibre-dominated configuration, similarly to the tensile case. However, the mechanisms governing failure under compressive loading differ significantly due to the instability phenomena that can develop within the fibre-matrix system.

Experimental investigations have been performed using Split Hopkinson Pressure Bar (SHPB) systems in order to analyse the compressive behaviour of unidirectional carbon-epoxy laminates at high strain rates [20]. Tests conducted at strain rates of approximately 100 s^{-1} show that the stress-strain response remains approximately linear within the elastic region. When comparing quasi-static and dynamic loading conditions, the longitudinal compressive modulus does not show a significant dependence on strain rate. In contrast, the compressive strength exhibits a noticeable increase with increasing strain rate, with some studies reporting an increase of approximately 40% when comparing quasi-static and dynamic regimes [20].

Failure analysis performed on dynamically tested specimens reveals that compressive failure is mainly governed by instability mechanisms within the fibre-matrix system. Scanning electron microscopy observations show that failure typically occurs through fibre microbuckling accompanied by splitting and delamination of the laminate. The formation of kink bands has also been reported, with kink-band widths ranging approximately between 8 and 16 fibre diameters [22].

These observations indicate that, although the elastic stiffness of unidirectional laminates loaded in compression remains only weakly affected by the strain rate, the compressive strength and failure mechanisms are strongly related to fibre instability and fibre-matrix interaction phenomena. The experimental results discussed in the previous sections highlight the strong influence of strain rate on the mechanical response and failure mechanisms

of CFRP laminates. In order to experimentally investigate the behaviour of materials under such dynamic loading conditions, specific testing techniques capable of reaching very high strain rates are required. Among these techniques, the Split Hopkinson Pressure Bar (SHPB) represents one of the most widely used experimental methods for the characterisation of materials under high strain rate loading conditions.

5.3 Split Hopkinson Pressure Bar Technique

One of the most widely used experimental techniques for the dynamic characterisation of materials is the Split Hopkinson Pressure Bar (SHPB) [23]. This method allows the investigation of material behaviour under high strain rate loading conditions, typically ranging from approximately 10 to 10^3 s^{-1} for compression tests and from about 1 to 10^2 s^{-1} for tensile tests.

The SHPB apparatus is mainly composed of a striker bar, an incident bar and a transmission bar. The specimen to be tested is placed between the incident and the transmission bars. A schematic representation of the SHPB experimental setup is shown in Figure 5.1.

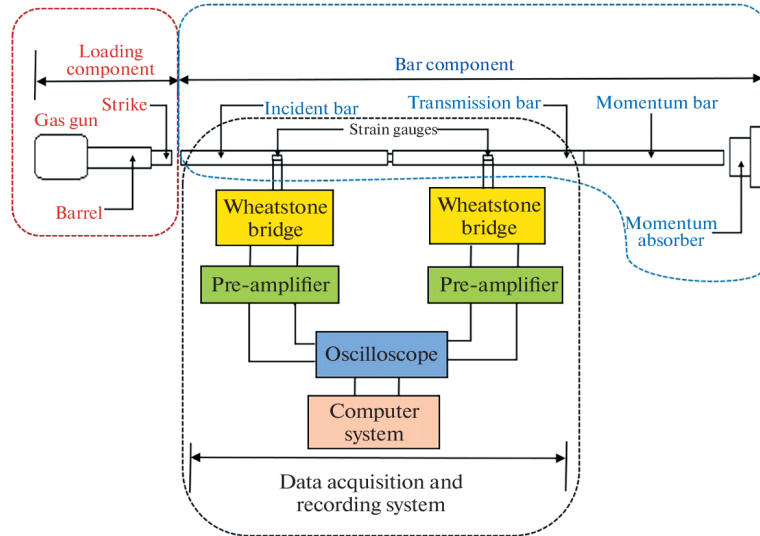


Figure 5.1: Schematic representation of a Split Hopkinson Pressure Bar (SHPB), reproduced from [23].

During the experiment, the striker bar is accelerated by a gas gun and impacts the incident bar at a known velocity. This impact generates a compressive stress wave that propagates along the incident bar towards the specimen. The stress wave generated in the incident bar is referred to as the *incident wave*.

When the incident wave reaches the interface between the incident bar and the specimen, part of the wave is reflected back into the incident bar due to the difference in mechanical impedance between the bar material and the specimen. This reflected component is called the *reflected wave*. The remaining portion of the wave propagates through the specimen and reaches the transmission bar, generating the *transmitted wave*.

Mechanical impedance represents the resistance of a material to the propagation of stress waves. The incident and reflected waves are measured using strain gauges mounted at the centre of the incident bar, while the transmitted wave is recorded by strain gauges positioned on the transmission bar. These signals are then analysed to determine the mechanical response of the specimen under high strain rate loading.

In order to obtain reliable experimental results, several fundamental conditions must be satisfied during SHPB testing. First, stress waves propagating within the bars must behave as one-dimensional elastic waves. Second, the striker, incident and transmission bars must remain within the elastic regime during the entire duration of the experiment. Third, the deformation of the specimen should remain as uniform as possible throughout the test. In addition, friction effects at the interfaces between the specimen and the bars should be minimised. Finally, the specimen must reach dynamic stress equilibrium before failure occurs in order to ensure the validity of the measured results [23].

5.3.1 Stress–Strain Determination in SHPB Tests

During a SHPB experiment, three strain signals are recorded on the pressure bars: the incident wave ε_I , the reflected wave ε_R , and the transmitted wave ε_T [23].

The amplitude of the stress wave is mainly governed by the impact velocity of the striker bar, while the duration of the loading pulse depends on the length of the striker bar. Based on the theory of one-dimensional elastic wave propagation and assuming dynamic equilibrium of the specimen during the test, the stress, strain and strain rate in the specimen can be expressed as

$$\sigma_c(t) = \frac{1}{2}E_b \left(\frac{A_b}{A_s} \right) (\varepsilon_I + \varepsilon_R + \varepsilon_T) \quad (5.1)$$

$$\varepsilon_c(t) = \frac{C_0}{L_0} \int_0^t (\varepsilon_I - \varepsilon_R - \varepsilon_T) dt \quad (5.2)$$

$$\dot{\varepsilon}_c(t) = \frac{C_0}{L_0} (\varepsilon_I - \varepsilon_R - \varepsilon_T) \quad (5.3)$$

where $\sigma_c(t)$ represents the stress history in the specimen, $\varepsilon_c(t)$ the strain history, and $\dot{\varepsilon}_c(t)$ the strain rate history. In addition, E_b is the elastic

modulus of the bars, A_b and A_s are the cross-sectional areas of the bar and the specimen respectively, C_0 is the elastic wave velocity in the bars, and L_0 is the initial length of the specimen.

Since the pressure bars are assumed to behave as linear elastic elements, the forces acting at the two interfaces of the specimen can be obtained from Hooke's law. The forces at the interfaces between the specimen and the incident and transmission bars can therefore be written as

$$F_1 = A_I E_I (\varepsilon_I + \varepsilon_R) \quad (5.4)$$

$$F_2 = A_T E_T (\varepsilon_T) \quad (5.5)$$

where F_1 represents the force at the interface between the specimen and the incident bar, while F_2 corresponds to the force at the interface between the specimen and the transmission bar. A_I and A_T denote the cross-sectional areas of the incident and transmission bars respectively.

Assuming that the specimen reaches dynamic stress equilibrium during the test and considering identical cross-sections for the incident and transmission bars, the forces at both interfaces must be equal:

$$F_1 = F_2 \Rightarrow \varepsilon_I + \varepsilon_R = \varepsilon_T \quad (5.6)$$

Under this equilibrium condition, the stress, strain and strain rate in the specimen can be simplified as

$$\varepsilon_c(t) = -\frac{2C_0}{L_0} \int_0^t \varepsilon_R(t) dt \quad (5.7)$$

$$\dot{\varepsilon}_c(t) = -\frac{2C_0}{L_0} \varepsilon_R(t) \quad (5.8)$$

$$\sigma_c(t) = E_b \left(\frac{A_b}{A_s} \right) \varepsilon_T(t) \quad (5.9)$$

These relations allow the reconstruction of the complete stress-strain response of the specimen under high strain rate loading conditions using the strain signals measured on the pressure bars.

Chapter 6

Conclusions

The main objective of this study was to characterize the interlaminar fracture behaviour of the investigated composite material through experimental testing and to assess the ability of numerical modelling to reproduce the observed mechanical response.

The experimental results provided a global description of the delamination behaviour of the analysed specimens. In particular, the fracture toughness values in Mode I and Mode II (G_{IC} and G_{IIC}) were identified from the tests and subsequently used as input parameters for the numerical simulations.

The numerical modelling carried out using Abaqus enabled a direct comparison between experimental and numerical force–displacement responses. Two different modelling approaches were considered, namely the Virtual Crack Closure Technique (VCCT) and the surface-based cohesive contact formulation. Both approaches were able to reproduce satisfactorily the linear elastic portion of the force–displacement curves. However, a less accurate agreement was observed in the region associated with the onset of instability and crack propagation.

Despite the limitations encountered, which are discussed in detail in the numerical modelling chapter, the combined use of experimental testing and numerical simulation provided a deeper understanding of the delamination behaviour of the investigated laminate.

Several possible developments can be identified for future work. First, additional experimental campaigns could be performed using modified test setups in order to improve the stability of crack propagation during the tests. From a numerical perspective, further improvements of the modelling strategies could allow a more accurate representation of the post-initiation phase of delamination.

Moreover, repeating the manufacturing process in order to obtain L-shaped specimens could enable the characterization of through-thickness stresses and interlaminar behaviour under different loading conditions. Fi-

nally, the experimental investigation could be extended to dynamic loading conditions, for instance by studying the response of the laminate at high strain rates using a Split Hopkinson Pressure Bar apparatus.

Appendix A

Additional Manufacturing Attempts

A.1 Manufacturing of an L-shaped specimen for through-thickness characterization

During the experimental campaign, an additional specimen was manufactured with the objective of characterizing the through-thickness strength of the composite laminate, corresponding to the stress component σ_{33} .

The goal of this specimen was to investigate the mechanical resistance of the laminate in the thickness direction. While the experimental tests presented in the previous chapters mainly focused on in-plane properties and interlaminar fracture behaviour, this specimen was designed to provide information about the out-of-plane strength of the material.

For this purpose, an L-shaped composite geometry was selected. This configuration allows the application of loads that generate significant stresses through the laminate thickness, making it suitable for investigating the σ_{33} component.

A.1.1 Specimen preparation

The specimen was manufactured using carbon/epoxy prepreg material. The prepreg plies were first cut according to the required dimensions while ensuring that the fibre orientation was aligned with the reference direction (0°).

The plies were then manually laid up on a metallic mould having an L-shaped geometry. This metallic tool imposes the final geometry of the specimen during the curing process.

Figure [A.1](#) shows the layup stage of the laminate on the metallic tool before the vacuum bagging configuration was completed.

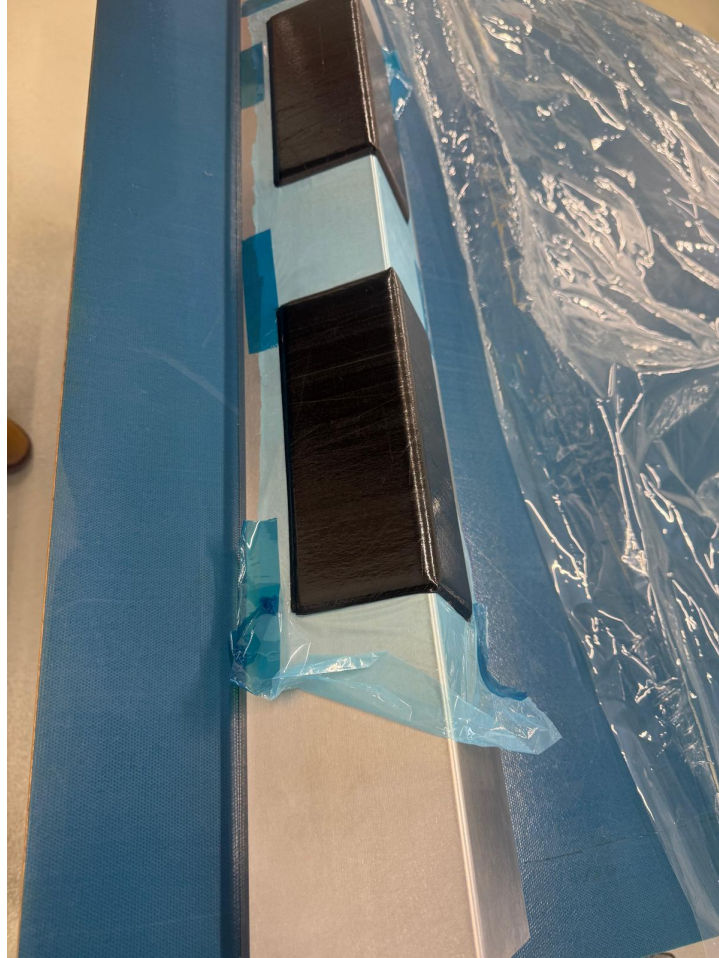


Figure A.1: Layup of the carbon/epoxy prepreg laminate on the L-shaped metallic tool.

A.1.2 Vacuum bagging configuration

After completing the layup of the laminate, the vacuum bagging configuration was prepared in order to consolidate the laminate and prepare it for the autoclave curing cycle.

The stacking sequence used during the vacuum bagging process, from the mould surface to the vacuum bag, was the following:

- metallic tool (L-shaped mould)
- Teflon release film
- peel ply
- carbon/epoxy laminate (C/E)

- peel ply
- perforated Teflon release film
- glass fibre absorber ply (dry)
- Teflon film
- drainant (breather) ply
- vacuum bag

The release film prevents the laminate from sticking to the metallic tool and facilitates demoulding after curing. The peel ply layers allow an easier separation between the composite laminate and the surrounding consumable materials. The perforated Teflon film controls the resin flow during the curing process, while the glass fibre absorber ply acts as a bleeder layer by absorbing the excess resin expelled during curing.

The drainant ply allows air evacuation and ensures a proper vacuum distribution inside the bag during the curing cycle.

Figure A.2 shows the complete vacuum bagging configuration before the curing cycle.



Figure A.2: Vacuum bagging configuration prepared before the curing cycle.

A.1.3 Schematic representation of the bagging stack

For clarity, a schematic representation of the vacuum bagging stack used during the manufacturing process is shown in Figure A.3.

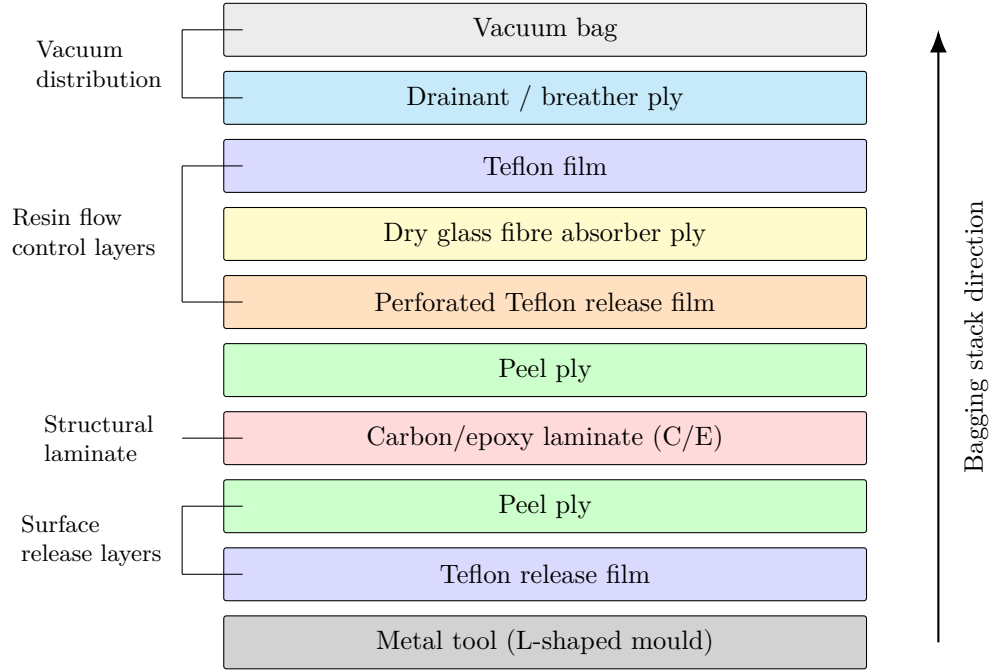


Figure A.3: Schematic representation of the vacuum bagging stack used during the manufacturing process.

A.1.4 Temperature monitoring

During the curing process, several thermocouples were installed in order to monitor the temperature evolution during the autoclave cycle. The thermocouples were positioned on the mould and close to the laminate in order to record the temperature history experienced by the specimen during curing.

This temperature monitoring was necessary to verify that the curing cycle followed the recommended curing profile of the prepreg system.

Figure A.4 shows the positioning of the thermocouples on the vacuum bagged specimen.

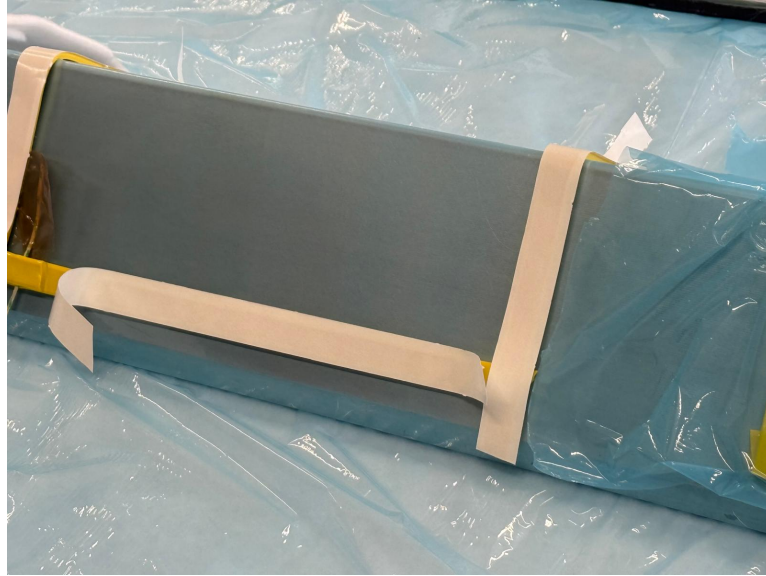


Figure A.4: Positioning of thermocouples used to monitor the temperature during the curing cycle.

A.1.5 Curing issue and testing limitation

During the autoclave curing cycle, the recorded temperature exceeded the value recommended for the prepreg system. In particular, the temperature reached values higher than 200°C, which is above the expected curing temperature range for the material.

This excessive temperature likely altered the curing behaviour of the resin system and negatively affected the final quality of the manufactured component.

The demoulded composite specimen is shown in [Figure A.5](#).



Figure A.5: Demoulded L-shaped composite specimen after the curing process.

Because of the temperature overshoot during the curing cycle, the resulting part did not present the required structural quality to be used for mechanical testing. For this reason, the specimen could not be used for the planned experimental characterization of the through-thickness strength σ_{33} .

Due to the limited time available at the end of the experimental campaign, it was unfortunately not possible to reproduce the manufacturing process and fabricate a new specimen. Consequently, the experimental characterization of the σ_{33} strength could not be carried out within the timeframe of this work.

Bibliography

- [1] A. U. Sudhin, Manu Remanan, G. Ajayesh, and Karingamanna Jayanarayanan. Comparison of properties of carbon fiber reinforced thermoplastic and thermosetting composites for aerospace applications. *Materials Today: Proceedings*, 24:453–462, 2020.
- [2] Dipen Kumar Rajak, Durgesh D. Pagar, Ravinder Kumar, and Catalin I. Pruncu. Recent progress of reinforcement materials: a comprehensive overview of composite materials. *Journal of Materials Research and Technology*, 8(6):6354–6374, 2019.
- [3] Hamza El Etri, Mehmet Ziya Polat, and Emin Burak Özyilmaz. Classification, applications, and impact behavior of composite materials: A review. *Mechanics of Advanced Materials and Structures*, 31(30):12592–12635, 2024.
- [4] Dipen Kumar Rajak, Durgesh D. Pagar, Pradeep L. Menezes, and Emanoil Linul. Fiber-reinforced polymer composites: Manufacturing, properties, and applications. *Polymers*, 11(10):1667, 2019.
- [5] K. M. Liew, Z. Z. Pan, and L. W. Zhang. An overview of layerwise theories for composite laminates and structures: Development, numerical implementation and application. *Composite Structures*, 216:240–259, 2019.
- [6] Ramesh Talreja and Chandra Veer Singh. Damage in composite materials. In *Damage and Failure of Composite Materials*, pages 36–56. Cambridge University Press, 2012.
- [7] T. E. Tay. Characterization and analysis of delamination fracture in composites: An overview of developments from 1990 to 2001. *Applied Mechanics Reviews*, 56(1):1–32, 2003.
- [8] H. Zabala, L. Aretxabaleta, G. Castillo, and J. Aurrekoetxea. Loading rate dependency on mode i interlaminar fracture toughness of unidirectional and woven carbon fibre epoxy composites. *Composite Structures*, 121:75–82, 2015.

- [9] S. K. Maiti. Linear elastic fracture mechanics. In *Fracture Mechanics: Fundamentals and Applications*, pages 6–64. Cambridge University Press, 2015.
- [10] V. Pavelko, K. Lapsa, and P. Pavlovskis. Determination of the mode i interlaminar fracture toughness by using a nonlinear double-cantilever beam specimen. *Mechanics of Composite Materials*, 52(3):347–358, 2016.
- [11] Frédéric Lachaud, Robert Piquet, Grégory Aldebert, Jacques Huet, and Laurent Michel. Analyse du comportement mécanique des assemblages boulonnés composites à renforts tissés. *Revue des composites et des matériaux avancés*, 24(4):449–464, 2014.
- [12] T. Andersson and U. Stigh. The stress–elongation relation for an adhesive layer loaded in peel using equilibrium of energetic forces. *International Journal of Solids and Structures*, 41(2):413–434, 2004.
- [13] Victor Médeau. *Rupture des composites tissés 3D : de la caractérisation expérimentale à la simulation robuste des effets d’échelle*. PhD thesis, Institut Supérieur de l’Aéronautique et de l’Espace (ISAE-SUPAERO), 2019.
- [14] M. K. Budzik, J. Jumel, N. Ben Salem, and M. E. R. Shanahan. Instrumented end notched flexure – crack propagation and process zone monitoring. part ii: Data reduction and experimental analysis. *Engineering Fracture Mechanics*, 210:197–215, 2019.
- [15] Astm d7905/d7905m – standard test method for determination of the mode ii interlaminar fracture toughness of unidirectional fiber-reinforced polymer matrix composites, 2019. Reapproved 2019.
- [16] Francesco Parrinello. Analytical solution of the 4enf test with interlaminar frictional effects and evaluation of mode ii delamination toughness. *Journal of Engineering Mechanics*, 144(4):04018011, 2018.
- [17] Mohammad Heidari-Rarani and Mousa Sayedain. Finite element modeling strategies for 2d and 3d delamination propagation in composite dcb specimens using vcct, czm and xfem approaches. *Theoretical and Applied Fracture Mechanics*, 103:102246, 2019.
- [18] Jordan Berton, Fabien Coussa, Julien Berthe, Eric Deletombe, and Mathias Brieu. Investigation of strain rate effects on the irreversible and damageable behavior of carbon fiber reinforced polymers under dynamic loading. *International Journal of Damage Mechanics*, 2025.

- [19] K. Naresh, K. Shankar, B. S. Rao, and R. Velmurugan. Effect of high strain rate on glass/carbon/hybrid fiber reinforced epoxy laminated composites. *Composite Structures*, 2015.
- [20] H. Koerber and P. P. Camanho. High strain rate characterisation of uni-directional carbon-epoxy im7-8552 in longitudinal compression. *Composites Part A: Applied Science and Manufacturing*, 42:462–470, 2011.
- [21] Donglin Gao, Zuguo Bao, Weijian Han, Xianpeng Wang, Shiyao Huang, Li Huang, Qiuren Chen, Hailong Zhao, and Yahong Xu. Effect of strain rate on tensile properties of carbon fiber-reinforced epoxy laminates with different stacking sequences and ply orientations. *Materials*, 2023.
- [22] S. Sivashanker, S. O. Osiyemi, and A. Bag. Compressive failure of a uni-directional carbon-epoxy composite at high strain rates. *Metallurgical and Materials Transactions A*, 34(6):1396–1399, 2003.
- [23] M. M. Khan and M. A. Iqbal. Dynamic increase factor of concrete subjected to compression and tension by using split hopkinson pressure bar setup: Overview. *Mechanics of Solids*, 58(6):2115–2131, 2023.