Computational Modelling of Failure of Stiffened Composite Panels

Gonçalo Miguel Ramalho Brás Pereira

Thesis to obtain the Master of Science Degree in Aerospace Engineering

Supervisors: Prof. Nuno Miguel Rosa Pereira Silvestre
Dr. António Pedro Carones Duarte

Examination Committee

Chairperson: Prof. Fernando José Parracho Lau
Supervisor: Prof. Nuno Miguel Rosa Pereira Silvestre
Members of the Committee: Prof. José Arnaldo Pereira Leite Miranda Guedes
Dr. António Pedro Carones Duarte

December 2017
Acknowledgments

This dissertation represents the end of a period of eight months that allowed me to grow not only as an engineer, but also on a personal level. I would like to reflect on the people who have supported and inspired me throughout this period.

Firstly, I would like to thank my family, which includes my father, mother and sister for supporting me during this entire process. Without them this thesis would not be possible.

Secondly, I must thank my girlfriend for being a constant source of motivation and strength. From now on I will have more time for you.

Thirdly, I would like to express my deepest gratitude to both my thesis’ supervisors, Professor Nuno Silvestre and Eng. António Duarte for their guidance, remarks and engagement through the learning process. I have learned many things since I met professor Nuno in the first class of computational mechanics and I thank him for all the knowledge he passed along to me. Special thanks to António Duarte for introducing me to the Abaqus software and for providing always useful suggestions and remarks.

Finally, I would like to dedicate this thesis to my grandmother, Madalena Ramalho, who recently passed away. I will be forever grateful for your presence in my life.
Abstract

This dissertation presents an in-depth computational study on the buckling, postbuckling and strength of stiffened composite panels. It follows up the finished COCOMAT project, supported by the European Commission, with the aim of exploiting large strength reserves in stiffened carbon fiber reinforced polymer (CFRP) fuselage structures. The main goals are to improve the structural efficiency and decrease the structural weight and development and operation costs.

Several Finite Element (FE) models were developed throughout this work and extensive simulations were carried out. The first numerical simulations comprised the postbuckling analysis of a thin-walled stiffened CFRP panel subjected to axial compression with T-shaped stringers, similar to that studied in the COCOMAT project. Alternative damage models considering strength-based criteria and fracture mechanics (Hashin, cohesive elements and eXtended FE Method (XFEM)) were implemented to capture intra-laminar damage in the composite and adhesive failure, respectively. Fiber failure and the detachment between the skin and stringers, caused by damage of the adhesive, were identified as the most severe damage mechanisms leading to structural collapse. Validation of the model of the first panel design arose from the good agreement obtained between the numerical and the experimental and numerical results obtained in the COCOMAT project.

Additional models of several panel designs with different stringer cross-section shapes were created to evaluate their structural behavior under axial compression and bending. The load/moment-carrying capacity and collapse of those panels were analyzed and compared. The one with Ω-shaped stringers revealed to be the most efficient, presenting the highest exploitation of postbuckling reserve strength and lowest weight, thus being recommended to be studied for possible future applications.

**Keywords:** Stiffened panel, structural efficiency, composite materials, computational analyses, buckling and postbuckling, damage mechanisms
Resumo

Esta dissertação apresenta um estudo aprofundado sobre o comportamento de estabilidade e pós-encurvadura de painéis compósitos. O mesmo surge no seguimento do projecto COCOMAT, apoiado pela Comissão Europeia, e visa ser uma contribuição directa para o estudo do aumento da eficiência estrutural de painéis fabricados em polímero reforçado com fibras de carbono (CFRP) utilizados em fuselagem de aviões. Para tal, pretende-se aumentar a exploração da sua resistência de pós-encurvadura e simultaneamente reduzir o peso estrutural e custos de desenvolvimento e operação.

Utilizou-se o software de elementos finitos Abaqus para estudar o comportamento estrutural de diversos painéis CFRP, representativos da secção de uma fuselagem, quando submetidos a compressão axial. O primeiro modelo desenvolvido visou reproduzir aquele estudado no projecto COCOMAT, apresentando reforços com secção transversal em "T". Para esta geometria, estudou-se a influência de diferentes modelos de dano intralamina, do compósito, e do adesivo no comportamento do painel, tendo sido utilizados o critério de Hashin, os elementos coesivos e o extende Finite Element Method (XFEM)). Verificou-se que o dano nas fibras do compósito e a separação entre a casca e os reforços, devida ao dano no adesivo, conduzem ao colapso repentino da estrutura. A validação do modelo fez-se por comparação dos resultados numéricos com os obtidos experimental e numericamente no projecto COCOMAT.

Foram ainda criados modelos de painéis com diferentes geometrias de secção dos reforços, estudando-se o seu comportamento à compressão e à flexão. O painel com o reforço com secção transversal em forma de "Ω" foi identificado como sendo o mais eficiente, apresentando a melhor exploração da resistência de pós-encurvadura e o menor peso estrutural.

Palavras-chave: Painéis compósitos reforçados, eficiência estrutural, análises computacionais, instabilidade e pós-encurvadura, mecanismos de dano
# Table of Contents

Acknowledgments ......................................................................................... iii  
Abstract ........................................................................................................ v  
Resumo .......................................................................................................... vii  
Table of Contents ........................................................................................ ix  
List of Figures ............................................................................................... xi  
List of Tables ................................................................................................ xv  
Notation ........................................................................................................ xvi  

1. Introduction ............................................................................................... 21
   1.1. Context .............................................................................................. 21
   1.2. Objectives ......................................................................................... 22
   1.3. Methods of analysis ......................................................................... 23
   1.4. Organization of the document ......................................................... 23

2. Literature Review ...................................................................................... 25
   2.1. FE tools and analysis procedures ..................................................... 25
   2.2. Elasticity of fiber-reinforced materials ............................................. 30
   2.3. Damage mechanisms and models ..................................................... 34
       2.3.1. Damage mechanisms ................................................................ 34
       2.3.2. Damage characterization and modelling ................................... 35
   2.4. Buckling, postbuckling and strength of stiffened panels .................. 44

3. FE Model ................................................................................................. 49
   3.1. Geometry ......................................................................................... 50
   3.2. FE mesh ........................................................................................... 52
   3.3. Material properties and damage models .......................................... 55
   3.4. Boundary and loading conditions .................................................... 59
   3.5. Methods of analysis ........................................................................ 61

4. Results and Discussion ............................................................................ 63
   4.1. Buckling of panels under compression ........................................... 63
   4.2. Postbuckling of reference panel T5 under compression ..................... 64
       4.2.1. Model without damage .............................................................. 64
4.2.2. Influence of imperfections and damping factor ........................................ 67
4.2.3. Models with damage ............................................................................. 69
  4.2.3.1 Hashin’s damage model ..................................................................... 69
  4.2.3.2 Hashin’s damage model with cohesive elements ............................... 74
  4.2.3.3 Hashin’s damage model with XFEM .................................................. 77
4.2.4. Comparison of results ........................................................................... 80

4.3. Postbuckling of different panels under compression .................................. 85

4.4. Postbuckling of panels under bending ......................................................... 90
  4.4.1. Reference panel T5 ............................................................................. 90
  4.4.2. Influence of stringer number and shape ................................................. 92

5. Conclusions and Future Developments .......................................................... 97
  5.1. Conclusions ............................................................................................. 97
  5.2. Future developments ............................................................................... 98

6. References .................................................................................................. 99

Appendix A .................................................................................................. 101
Appendix B .................................................................................................. 106
Appendix C .................................................................................................. 109
List of Figures

Figure 1: Visual representation of a fuselage stiffened panel [3] ................................................................. 22
Figure 2: First iteration of an increment in Newton’s Raphson method [5] ......................................................... 28
Figure 3: Second iteration of an increment in Newton’s Raphson method [8] ..................................................... 29
Figure 4: Homogenisation of ply properties in a single lamina [4] ................................................................. 30
Figure 5: Local coordinate system of a single lamina [8] .................................................................................. 31
Figure 6: Local and global coordinate system of a single layer [9] .................................................................. 33
Figure 7: Lever arms of single layers of the laminate [8] ................................................................................. 33
Figure 8: Damage mechanisms in composite stiffened structures .................................................................. 34
Figure 9: Crack growth modes [17] ............................................................................................................. 36
Figure 10: Equivalent stress versus equivalent displacement [4] ................................................................. 39
Figure 11: VCCT method for pure Mode I [24] ......................................................................................... 41
Figure 12: Traction-separation cohesive behaviour ....................................................................................... 42
Figure 13: Element removal using cohesive elements [25] ........................................................................ 44
Figure 14: Definition of local and global buckling, and collapse [2] ............................................................. 45
Figure 15: Load-shortening curve of the experimental test [4] ...................................................................... 45
Figure 16: Experimental deformation patterns at different axial shortenings (u): The red/yellow colours represent outward displacement and the blue/green inward displacement (adapted from [8]) ........... 46
Figure 17: Load-shortening curve of different subroutines in comparison to the experiment ................. 46
Figure 18: Failure propagation of the adhesive layer at 4 load levels: A-70 kN; B-75 kN; C-79 kN; D-83 kN ................................................................................................................................. 47
Figure 19: Global deformation pattern at collapse for different versions of the USDFLD subroutine. The red/yellow colours represent outward displacement and the blue/green inward displacement (adapted from [8]) ................................................................................................................ 47
Figure 20: Analysis procedure in Abaqus .................................................................................................. 49
Figure 21: Geometry and dimensions of the t-shaped stringers used in the COCOMAT project [8] .... 50
Figure 22: Midsurface shell approach for the T-shaped stringers ............................................................. 51
Figure 23: C3D8 elements with full integration [4] ........................................................................................ 53
Figure 24: Convergence study with two mesh variations ........................................................................... 55
Figure 25: Assembly of the model with DM-HC ...................................................................................... 60
Figure 26: Viewport of the final mesh with the reposition/adjust of the slave nodes (DM-HC) ........... 60
Figure 27: Boundary conditions applied (general panel) [2].......................................................... 60
Figure 28: Load-shortening curve for panel design T5 without damage ........................................... 65
Figure 29: Load-shortening curve of the panel design T5 without any type of damage ................. 66
Figure 30: Influence of the geometrical imperfections in the nonlinear analysis using Abaqus/Standard ........................................................................................................................................ 67
Figure 31: Influence of the damping factor in the nonlinear analysis using Abaqus/Standard ....... 68
Figure 32: Load-shortening curve for panel design T5 with DM-H .................................................. 69
Figure 33: Damage initiation (matrix crushing mode) for the 90º layers of the skin matrix in (a) bottom ply and (b) top ply for uz =1.0 mm ........................................................................................................... 70
Figure 34: Degree of damage (matrix crushing mode) for the (a) bottom ply and (b) top ply of the skin for uz = 1.05 mm ........................................................................................................................................... 70
Figure 35: Degree of damage (matrix crushing mode) for the (a) bottom and (b) top plies of the skin for uz = 1.17 mm (global buckling) ........................................................................................................................................... 71
Figure 36: Out-of-plane displacements (y-direction) at uz = 1.17 mm (global buckling) ............ 71
Figure 37: Degree of damage (matrix crushing mode) for the 0º plies of the stringer-flange for uz = 1.8 mm ........................................................................................................................................... 72
Figure 38: Fibre kinking damage variable for one 0º ply of the stringer-blade at an axial shortening of 2.5 mm ........................................................................................................................................... 73
Figure 39: Fibre kinking damage variable for one -45º ply of the stringer-flange for uz = 3.3 mm .... 73
Figure 40: Fibre breakage in the topmost ply of the skin underneath the central stiffener for uz = 3.27 mm ........................................................................................................................................... 74
Figure 41: Load-shortening curve of the model with cohesive elements in the adhesive layer (DM-HC) ........................................................................................................................................... 75
Figure 42: First failed adhesive elements for uz = 2.26 mm ........................................................... 76
Figure 43: Failed elements in the adhesive / debonded areas for uz = 2.36 mm ......................... 76
Figure 44: Load-shortening curve of the model with XFEM (DM-HX) ............................................ 78
Figure 45: Fiber kinking in the stringer blades for uz = 2.55 mm (top) and uz = 2.65 mm (bottom) .... 78
Figure 46: Multiple cracks at the adhesive layer (stringer 5 is not visible) ................................. 79
Figure 47: Details (zooms) of crack propagation in the vicinity of the adhesives underneath the stiffeners 3 and 4 ........................................................................................................................................... 79
Figure 48: Load-shortening curves of the numerical models developed and experiment ............ 80
Figure C-4: Degree of damage (fiber kinking mode) for one ply of the stringer blade oriented at 0° (bottom) and at 45° (top) for uz = 3.3 mm .................................................................112

Figure C-5: Fiber kinking in stringer blade 2 for uz = 2.36 mm .................................................................112

Figure C-6: Second global buckling shape developed in the vicinity of two of stringers 3 and 4 from uz = 1.95 mm to uz = 2.05 mm .................................................................113

Figure C-7: Matrix crushing (left) and cracking(right) at the bottom ply of skin and stringers after collapse.........................................................................................................................113

Figure C-8: Load-shortening curve of two versions of panel design T5.........................................................117

Figure C-9: Load-shortening curve of two versions of panel design T4 ...........................................................117

Figure C-10: Load-shortening curve of two versions of panel design T6 .........................................................118

Figure C-11: Load-shortening curve of two versions of panel design I .........................................................118

Figure C-12: Load-shortening curve of two versions of panel design C .........................................................118

Figure C-13: Load-shortening curve of two versions of panel design J .........................................................119

Figure C-14: Degree of damage (matrix crushing mode) for the top ply of the skin for θy = 5.86 x10⁻³ rad.................................................................119
List of Tables

Table 1: Geometric data of the Panel T5........................................................................................................50
Table 2: Geometry and dimensions of the panel designs I, C, J and Ω.........................................................52
Table 3: FE model parameters of all designs (Panel T5) .............................................................................54
Table 4: Material properties for CFRP prepeg IM7/8552 UD [8] .............................................................56
Table 5: Material data of the adhesive Redux 312 [2] ..................................................................................56
Table 6: Orientations and laminate layups of the skin and stringers..............................................................57
Table 7: Damage models................................................................................................................................58
Table 8: Fracture energies of the CFRP IM7/8552 [5] and of the adhesive Redux 312 [27] ..............58
Table 9: Incrementation parameters..............................................................................................................62
Table 10: Results of the linear analysis all panel designs ............................................................................64
Table 11: Typical behaviour of stringer stiffened panels and presentation of 3 load levels......................66
Table 12: Deformation patterns at different values of axial shortening......................................................83
Table 13: Comparison of results between the 7 panels ..............................................................................86
Table 14: Comparison of results between the 7 panel designs (bending) ..................................................93
Table A-1: Output variables related to damage initiation and evolution in FRP composites ..............102
Table A-2: Output variables related to cohesive elements .........................................................................103
Table B-1: Mesh densities of all panel designs and respective illustration ..........................................107
Table C-1: First eigen shapes of the several panel designs ......................................................................109
Table C-2: Deformation patterns of all panel designs. .............................................................................114
### Notation

<table>
<thead>
<tr>
<th>Term</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>Arc length</td>
</tr>
<tr>
<td>b</td>
<td>Stringer width</td>
</tr>
<tr>
<td>$c_a$, $c_b$</td>
<td>Displacement correction (Section 2.1.4)</td>
</tr>
<tr>
<td>$C_d$</td>
<td>Damage elasticity matrix</td>
</tr>
<tr>
<td>d</td>
<td>damage variable for a particular mode</td>
</tr>
<tr>
<td>$d_f$, $d_m$, $d_s$</td>
<td>Internal variables that characterize fiber, matrix and shear damage</td>
</tr>
<tr>
<td>$d_v$</td>
<td>Damage variable with the viscous regularization scheme applied</td>
</tr>
<tr>
<td>E</td>
<td>Young's modulus</td>
</tr>
<tr>
<td>$F_f^I$, $F_f^C$, $F_m^I$, $F_m^C$</td>
<td>Hashin's damage initiation criteria</td>
</tr>
<tr>
<td>$F_v$</td>
<td>Viscous forces</td>
</tr>
<tr>
<td>G</td>
<td>Shear modulus</td>
</tr>
<tr>
<td>$G_i$</td>
<td>Energy release rate</td>
</tr>
<tr>
<td>$G^c$</td>
<td>Critical energy release rate/ fracture energy</td>
</tr>
<tr>
<td>h</td>
<td>Stringer distance</td>
</tr>
<tr>
<td>$I$, $I_a$</td>
<td>Internal forces (Section 2.1.4)</td>
</tr>
<tr>
<td>$K_0$, $K_a$</td>
<td>Stiffness matrix of the structure (Section 2.1.4)</td>
</tr>
<tr>
<td>L</td>
<td>Panel length</td>
</tr>
<tr>
<td>$L_c$</td>
<td>Characteristic length</td>
</tr>
<tr>
<td>$M$</td>
<td>Damage operator matrix</td>
</tr>
<tr>
<td>$M$</td>
<td>Bending moment</td>
</tr>
<tr>
<td>$M_u$</td>
<td>Maximum bending moment with respect to the y-axis that the panel can withstand</td>
</tr>
<tr>
<td>$M^*$</td>
<td>Artificial mass matrix</td>
</tr>
<tr>
<td>$m_x$, $m_y$, $m_{xy}$</td>
<td>Moment resultants</td>
</tr>
<tr>
<td>$n_x$, $n_y$, $n_{xy}$</td>
<td>Force resultants</td>
</tr>
<tr>
<td>P</td>
<td>External force</td>
</tr>
<tr>
<td>$P_u$</td>
<td>Maximum compressive force transmitted through the panel</td>
</tr>
</tbody>
</table>
Radius

Residual forces (Section 2.1.4)

Longitudinal shear strength

Transverse shear strength

Maximum values of the nominal stress in the cohesive layer.

Ply thickness

Displacements (Section 2.1.4)

Strength in fiber direction

Strength in transverse direction

Greek letters

Coefficient related to Hashin's initiation criterion (Section 2.3.3)

Nodal enriched degrees of freedom (Section 2.3.3.2)

Gap opening (Section 2.3.3.4)

Vector of nodal velocities (Section 2.1.4)

Poisson's ratio

Equivalent displacements (Section 2.3.3.1)

Ply angle

Rotation around y-axis

Normal stress

Shear stress

Strength limit (Section 2.3.3.4)

Strain

Viscosity coefficient

Increment

Shortening between the first local and global buckling

Shortening between the global buckling load and collapse

Postbending Rotation
Subscripts/Superscripts

∥ Parallel direction
⊥ Perpendicular direction
1,2,3 In ply coordinate system: fiber, matrix and through-thickness directions
I, II, III Crack opening modes
c, C Compression, used with material strength data
f, m Fiber, matrix
nn, s₁, s₂ Normal and the two shear directions in the cohesive layer
t, T Tensile, used with material strength data

Acronyms and abbreviations

BC Boundary conditions
BK Benzeggagh-Kenane
CDM Continuum damage mechanics
CFRP Carbon fiber-reinforced polymers
CLPT Classic laminated plate theory
D1 Design 1
DCB Double cantilever beam
DF Damping factor
DLR Deutsches Zentrum fur Luft-und Raumfart
DM Damage model
DOF Degrees of freedom
ENF End notched flexure
ERR Energy release rate
FE Finite element
FEM Finite element method
FEA Finite element analysis
FLFS Fist loss of flexural stiffness
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FRP</td>
<td>Fiber-reinforced plastic</td>
</tr>
<tr>
<td>LEFM</td>
<td>Linear elastic fracture mechanics</td>
</tr>
<tr>
<td>MAXS</td>
<td>Maximum nominal stress criterion</td>
</tr>
<tr>
<td>MTS</td>
<td>Maximum tangential stress</td>
</tr>
<tr>
<td>PDEs</td>
<td>Partial Differential Equations</td>
</tr>
<tr>
<td>Prepeg</td>
<td>Pre-impregnated</td>
</tr>
<tr>
<td>ODB</td>
<td>Output database</td>
</tr>
<tr>
<td>QUADS</td>
<td>Quadratic nominal stress criterion</td>
</tr>
<tr>
<td>RF</td>
<td>Reaction Force</td>
</tr>
<tr>
<td>RP</td>
<td>Reference point</td>
</tr>
<tr>
<td>TSL</td>
<td>Traction-separation law</td>
</tr>
<tr>
<td>UD</td>
<td>Unidirectional</td>
</tr>
<tr>
<td>USLFLD</td>
<td>User-defined field</td>
</tr>
<tr>
<td>VCCT</td>
<td>Virtual Crack Closure Technique</td>
</tr>
<tr>
<td>XFEM</td>
<td>eXtended Finite Element Method</td>
</tr>
<tr>
<td>1D</td>
<td>One dimensional</td>
</tr>
<tr>
<td>2D</td>
<td>Two dimensional</td>
</tr>
<tr>
<td>3D</td>
<td>Three dimensional</td>
</tr>
<tr>
<td>max</td>
<td>Maximum</td>
</tr>
</tbody>
</table>
1. Introduction

The application of composite materials in the aerospace industry has been widespread for decades. The main concerns have been related to the durability of composite structures, as well as to the accuracy of design tools. However, in recent years, advanced carbon fiber-reinforced polymers (CFRP) are increasingly being introduced into primary fuselage aircraft and spacecraft structures, as engineers are always striving for improving performance and structural efficiency, whilst reducing emissions and weight.

The design of fuselage structures taking into account their postbuckling strength has emerged throughout the years. These structures can carry high loads even after their initial buckling loads have been exceeded. Postbuckling-based design has successfully been applied to metallic aircraft structures, but its application with composites has been limited to date. In combination with the high performance of composite materials, the concept of postbuckling-based design has the potential to improve significantly the structural efficiency, since the ultimate loads can be increased by allowing the structures to be operated past the buckling points. Additionally, composite fuselage structures are lighter, which goes along with the continuous demand for cost reduction.

This new generation of composite fuselage structures requires a reliable and accurate simulation of postbuckling and collapse. Under compression, these structures experience buckling, adopt specific mode shapes and develop a wide range of damage mechanisms, which under further compression into the deep postbuckling region can lead to the collapse of the structure. The development of intra-laminar failure (ply damage) is critical to the collapse of composite structures. Delamination is a form of inter-laminar damage that significantly contributes to the loss of load-carrying capacity of composite laminates as well. However, for fuselage stiffened composite structures, the initiation of separation between the skin and the stringers (skin-stringer debonding) is typically the most sudden (abrupt) event causing collapse. As a result, engineers have been focused on modeling failure in stiffened composite structures over the years.

1.1. Context

Though experimental tests and numerical simulations have been performed on buckling and postbuckling of flat stiffened composite panels, on the other hand, studies on stiffened composite shells and stiffened composite curved panels were scarce at the starting time of the POSICOSS project ("Improved POSTbuckling SImulation for Design of Fibre COMposite Stiffened Fuselage Structures") [1] and its successor COCOMAT ("Improved MATerial Exploitation at Safe Design of COMposite Airframe Structures by Accurate Simulation of COLLapse") [2].

The European aircraft industry has demanded a reduction of the development and operating costs by 20% to 50% in the short and long terms, respectively. The COCOMAT project [2], which was comprised of 15 European partners and co-ordinated by the DLR (German Aerospace Center), started
in 2004 and ended in 2008 with the aim of improving the prediction of failure in the postbuckling region of stiffened composite curved panels, which are understood as parts of a fuselage section as depicted in Figure 1. COCOMAT benefited from the fast and reliable procedures developed by the POSICOSS team, which equivalently investigated the behaviour of stiffened composite panels under compression, but did not take material damage into account. Furthermore, the COCOMAT project went beyond the POSICOSS project by a simulation of structural collapse.

The COCOMAT partners developed subroutines programmed in FORTRAN to be used with Abaqus to model adhesive failure and skin-stringer separation, using simple stress-based failure criteria. However, the simulated models did not yield a realistic prediction of failure, as explained in greater detail in Chapter 2 (COCOMAT outcomes). Therefore, further improved damage models were needed at the end of the COCOMAT project.

The COCOMAT partners developed subroutines programmed in FORTRAN to be used with Abaqus to model adhesive failure and skin-stringer separation, using simple stress-based failure criteria. However, the simulated models did not yield a realistic prediction of failure, as explained in greater detail in Chapter 2 (COCOMAT outcomes). Therefore, further improved damage models were needed at the end of the COCOMAT project.

1.2. Objectives

The goals of this work can be divided into three main sections. The first one is to design and study the postbuckling behaviour of a thin-walled stiffened CFRP panel, which is assumed to be representative of a fuselage section, comprising T-shaped stringers with the same dimensions and material properties as those of the panel studied in the COCOMAT project. Alternative damage models for composite damage mechanisms and adhesive failure will be implemented. Prior to the incorporation of damage of the materials in the simulations, a numerical validation of the results will be necessary. Moreover, comparisons with the experimental results will be made.

The second purpose of this work will be to design different panel configurations and to carry out similar postbuckling analyses of the panels in order to evaluate the influence of the stringer geometry on the behavior of the panels under axial compression. Furthermore, the more efficient panel design will be pointed out regarding the exploitation of postbuckling reserve strength.

The load case considered for all investigations conducted in the COCOMAT project was axial compression under static loading. However, it is known that the axial stresses developed along the
circumferential direction of a fuselage are not uniform and may vary linearly. Therefore, the stress gradient arising from these 2nd order forces is well defined through a linear stress distribution equivalent to the application of a bending moment. Finally, the last goal of this work will be to incorporate an additional bending analysis to all panel configurations and draw some conclusions about the results (buckling, strength and failure mechanisms).

1.3. Methods of analysis

The numerical simulations carried out in this work incorporated the non-linear geometric behavior of both the skin and the stringers into account. Hence, firstly, a linear buckling/eigenvalue analysis was performed to extract buckling loads and modes and the latter were then used in the nonlinear simulations as artificial initial imperfections.

For the nonlinear simulations, a considerable number of parameters, such as (i) imperfections, (ii) skin-stringer connection, (iii) stringer-flange modelling, (iv) numerical solvers and damping factors applied, (v) mesh density and (vi) finite element (FE) types, were examined. The nonlinear analysis of all the panels was performed with the implicit solver provided in Abaqus/Standard [4] based on Newton’s Raphson method.

To represent the critical damage mechanisms of composite stiffened structures, three different damage models were studied, which can be considered as an alternative to the subroutines, which are very time-consuming when connected to Abaqus. These three modelling approaches are nowadays available within the Abaqus code and two of them were introduced by the time or after the COCOMAT project ended, namely (i) the new cohesive element technology (since Abaqus version 6.5) and (ii) the new ability to model crack propagation without the need for the crack to match the mesh, by means of the eXtended Finite Element Method (XFEM) framework (since Abaqus version 6.9). The other damage model approach is based on Hashin’s failure criteria for unidirectional fiber-reinforced composites.

1.4. Organization of the document

This document is arranged as follows. In this first Chapter, a brief introduction of the work as well as the presentation of the aims and scope is made. In Chapter 2, a literature review on three topics relevant to the analysis of composite stiffened structures is given, which covers some of the commercial nonlinear FE tools provided in ABAQUS, the principals of fiber-reinforced composite materials and a description of the damage models implemented in the FE models. Moreover, a review on the major outcomes of the COCOMAT project is also presented, which includes the results of the numerical simulations and of the available experimental data.

In Chapter 4, the numerical models developed for the purpose of this thesis are described in detail and Chapter 5 presents the numerical results and discussion. Finally, the overall conclusions of this work are shown, as well as the recommendations for further work regarding numerical simulations with different load cases and improved damage models.
2. Literature Review

This chapter presents a literature review on the main topics studied in detail throughout this project. The first one shows the major features of Abaqus/Standard [4], the commercial FE tool used in this work, and gives a description of the analysis procedure in Abaqus for solving nonlinear problems. Then, the principals of fiber-reinforced composite materials are described, which also includes a definition on linear elasticity. Afterwards, an introduction to the principal damage mechanisms in stiffened composite structures and the damage models applied in the numerical analysis of this work are presented. Finally, a brief review of the COCOMAT project outcomes is shown as the numerical models of panels studied in detail the next chapters of the thesis are initially calibrated by comparison with the results obtained in that investigation.

2.1. FE tools and analysis procedures

This section summarizes the commercial nonlinear FE tool Abaqus/Standard and the solution methods relevant to capture the structural behavior of a stiffened composite fuselage panel. Moreover, a description of the analysis procedure in Abaqus and a comparison between linear and nonlinear analysis are also given.

The deformation of a structure can be analytically described by a set of partial differential equations (PDEs). “Exact analytical solutions for the structural equations are available for only the most basic boundary and loading conditions, and likewise the application of analytical approximation techniques such as the Rayleigh-Ritz and Galerkin methods becomes prohibitively difficult for complicated geometries” [5]. Hence, for a computer to solve these PDEs, numerical techniques have been developed over the last few decades and the most powerful one for solving geometrically complicated structural problems is the FE Method (FEM).

The FEM was developed originally for the analysis of aircraft structures. The discretization applied in the FEM, according to Orificii [5], “allows for complex, nonlinear and history-dependent of all model parameters, including loads, boundary conditions, geometries, material properties and structural interactions such as contact.” The reader is referred to texts such as Ochoa and Reddy [6] for a more thorough description and theoretical development of this method. To conclude, the FEM is almost exclusively the numerical method used to capture the structural behavior of postbuckling composite structures.

In structural mechanics, the assumption of linearity is a simple mathematical approximation to simplify real-time problems. In linear analysis:

- the deflections and rotations are very small;
- stresses are proportional to strain;
- the equilibrium equations are written for the initial structural configuration;
- the stiffness matrix is constant;
- the material is considered to be linear elastic;
• the global equations (KU=P) are solved in a single step.

Though running linear FE analysis brings some advantages like short computing time, no convergence problems and an easy definition of the problem, in case where the linear behavior cannot be assumed, the outcome may be very wrong. In several real-time problems, all linearity assumptions cannot be satisfied, especially when significantly high loads are applied, and thus linear analysis may lead to inaccurate results.

In a nonlinear analysis, displacements vary non-linearly with the applied loads and changes in geometry cannot be ignored. The outcome is more robust, and it gives the actual deformation of the body. However, a nonlinear analysis requires much more computing time, it is far more difficult to set up and convergence problems may appear during the analysis process.

A high degree of nonlinearity is present in composite stiffened structures, where certain types of nonlinearity act simultaneously. The postbuckling analysis of this structures under compression involves large strains/rotations and thus nonlinear strain measures and kinematics must be taken under consideration (geometric nonlinearity). Additionally, Orificii [5] claimed that the compression of composite stiffened structures results in several damage mechanisms that represent nonlinearities, such as the reduction in material properties resulting from ply damage mechanisms or the loss of contact caused by the separation/debonding between the skin and stiffeners, as well as the potential delamination between the composite plies (contact nonlinearity) [4] [7].

To conclude, nonlinear FE analysis (FEA) is more expensive than linear FEA, but is necessary to efficiently capture the structural behavior of the stiffened composite fuselage panel due to the presence of a high degree of nonlinearity.

The application of the FE to study the postbuckling of structures requires the application of four stages [8]:

**Pre-processing:**
In this primary stage, the overall geometry of the FE model is created, followed by the material assignment to the different sections and parts of the model. After that, the FE mesh is created by choosing appropriate elements, regarding type (solid, shell, continuum shell, membrane, etc.) and dimensions.

**Linear eigenvalue analysis:**
This type of analysis is performed in order to extract buckling modes, which are subsequently used as initial imperfections in the nonlinear analysis [2]. These imperfections are “geometrical deviations from the perfect structural shape, which can occur randomly and unavoidably during manufacture” [5]. The introduction of imperfections in the FE model can slightly alter the deformation pattern and can even lead to the development of entirely different mode shapes.

**Nonlinear analysis:**
In this step, a solver is chosen to compute the unknown variables of the problem. The structural equations for any structure can be solved using either explicit or implicit solvers. An explicit FEA (given
in the FE tool Abaqus/Explicit) requires no iterations and no tangent stiffness matrix, as the internal force vector is assembled from contributions from the individual elements such that a global stiffness matrix need not be formed [4]. If the increments are small enough, accurate results will be computed, otherwise the solution will diverge. On the other hand, the implicit approach in Abaqus/Standard needs constant updating of the global stiffness matrix and a series of iterations to achieve convergence. In each increment, the Newton-Raphson method is applied to iterate the equilibrium. In this work, Abaqus/Standard was used, as it is more efficient for solving smooth nonlinear static problems. However, it is also numerically more expensive.

**Post-processing:**
This final stage consists in the visualization of the results that were calculated by the solver in the analysis. For this process, the output variables that are required for visualization must be previously defined (stresses, strains, displacements, reaction forces, velocities, energies, etc.). The graphic results are written in an output database (ODB) file.

Abaqus/Standard employs solution technology ideal for the analysis of the following events [4]:

- Linear and nonlinear static;
- Linear dynamic;
- Low speed nonlinear dynamic and quasi-static.

Abaqus/Standard uses the Newton-Raphson method to solve nonlinear equilibrium equations. In a nonlinear analysis, the solution cannot be calculated by solving a single system of linear equations. Instead, the simulation is broken into several time increments, and at the end of each one the approximate equilibrium configuration is found. The description of this method is given in Abaqus analysis user manual [4] and it involves a combination of incremental and iterative procedures. It is described in the following paragraphs.

A step consists of an analysis procedure, loading and output requests. Each step is broken into a certain number of increments so that it is possible to follow the nonlinear solution path. The user suggests the first increment, and Abaqus/Standard automatically chooses the size of the subsequent increments using automatic incrementation control. Within the increment, an iteration is an attempt to find an equilibrium solution. If the model is not in equilibrium at the end of the iteration, Abaqus tries another iteration, so that at the end of each increment the structure is in approximate equilibrium. The number of iterations needed to find a converged solution will vary depending on the degree of nonlinearity of the model. Sometimes the iteration process may diverge, so that subsequent iterations move further away from the equilibrium state. In this case the iteration process is terminated and Abaqus restarts the increment with a reduced increment size. This procedure is called “cut-back”.

**Convergence:**
For a body to be in equilibrium, the net force acting on every node must be zero, that is the external forces \( P \) and the internal forces \( I \) must balance each other:

\[
P - I = 0 \quad (2-1)
\]
The internal forces are caused by the stresses in the element that are attached to that node, whilst $P$ is the external load. The nonlinear response of a structure to a small load increment, $\Delta P$, is shown in Figure 2.

The stiffness matrix of the structure, $K_0$, which is based on its configuration at the displacement $u_0$, and the load increment, $\Delta P$, is used by Abaqus to calculate the displacement correction $c_a$. Using $c_a$, the structure’s configuration is updated to $u_a$, the total displacement in the increment. Abaqus then calculates the internal force $I_a$ in this updated configuration.

The difference between the total applied load, $P$, and the internal load $I_a$ is called the residual force for the iteration, $R_a$:

$$R_a = P - I_a$$  \hspace{1cm} (2-2)

If $R_a$ is zero at every degree of freedom (DOF), point a in Figure 2 would lie on the load-deflection curve, which means that the structure would be in equilibrium. When solving nonlinear problems $R_a$ will never be exactly zero, so Abaqus compares it to a tolerance value. If $R_a$ is less than the current tolerance value, the solution is accepted as being in equilibrium and $u_a$ is the valid equilibrium configuration for the structure under the applied load. On the other hand, Abaqus also checks that the last displacement correction, $c_a$, is small in comparison to the total increment displacement, $\Delta u_a = u_a - u_0$. In the case that $c_a$ is greater than a fraction of 1 % (by default) of the total increment displacement, another iteration is performed. When the two criteria are satisfied, the solution is converged for the increment and it is accepted by Abaqus.

If the solution from an iteration is not converged, another iteration is performed to bring the internal and external forces into balance. This iteration is shown in Figure 3.
First, the new stiffness matrix $K_a$ is calculated for the structure based on the updated configuration, $u_a$. The new stiffness and the residual force $R_a$ are then used to calculate a new displacement correction $c_b$, that brings the system closer to equilibrium (point b in Figure 3). After that, a new residual force $R_b$ is derived using the new internal forces from the structure’s new configuration $u_b$. Again, the residual force is compared to the force residual tolerance as well as the displacement $c_b$ is compared to the increment of displacement $\Delta u_b$. When these two criteria are not satisfied, Abaqus will perform further iterations.

In nonlinear problems with a large number of elements, the application of this method to the entire system of equations can be very time consuming, especially when the increment size is small and a lot of cut-backs are allowed. On the other hand, nonlinear static problems can be unstable. Such instabilities, which may be of geometrical nature or caused by material damage, manifest itself in a global load-displacement response with a negative stiffness or can be localized. In this last case, strain energy will be transferred from one part of the model to the adjoining parts. Therefore, the problem must be solved dynamically or with the aid of artificial damping (by using dashpots for example).

**Automatic stabilization of unstable problems**

Abaqus/Standard provides a mechanism for stabilizing unstable static or quasi-static problems through the addition of artificial damping. Viscous forces, $F_v$, are added to the global equilibrium equations:

$$ F_v = c M^* v \quad (2-3) $$

$$ P - I_a - F_v = 0 \quad (2-4) $$

with $v = \frac{\Delta u}{\Delta t}$. $M^*$ represents the artificial mass matrix calculated with the unit density, $c$ is the damping factor (DF), $v$ is the vector of nodal velocities and $\Delta t$ is the time increment (which may not have a physical meaning in the context of the problem).
As long as the model is stable, the viscous forces as well the viscous energy dissipated are very small, so the artificial damping does not affect the analysis. However, if a local region loses its stability, the local velocities increase and part of the released strain energy is dissipated by the applied artificial damping. Abaqus/Standard sets a value of $2.0 \times 10^{-4}$ for the DF automatically. It can be changed by the user by assigning the needed value from the automatic stabilization field.

2.2. Elasticity of fiber-reinforced materials

This section points out the main characteristics of fiber-reinforced materials and gives a brief overview of the Classic Laminated Plate Theory (CLPT) commonly used in the elastic analysis of composite materials. This theory is the fundamental theory underlying the analysis of fiber-reinforced composites and describes the assembly of a finite number of elastic orthotropic layers, or plies, into a total laminate, or plate [5]. Each ply is constructed by embedding many fibers in a matrix material. The combination of these two materials on a macroscopic scale provides better engineering properties than the conventional materials, like stiffness, strength, weight reduction, corrosion resistance, thermal properties and fatigue life [9].

To describe the mechanical behavior of a single unidirectional (UD) lamina, it is assumed that [9] [8]:

1) a lamina behaves as a linear elastic material;
2) a lamina is homogenous, which means that the individual fibres and matrix are not separately modelled, but are accounted for by “smearing” their properties into an orthotropic lamina. This means that the material has the same properties at every point. This is illustrated in Figure 4.

![Figure 4: Homogenisation of ply properties in a single lamina [4]](image)

To define the constitutive equations of a single UD lamina, an orthogonal local coordinate system is often used, which has the 1-axis aligned with the fiber direction, the 2-axis in the same plane but perpendicular to the fibers and finally the 3-axis perpendicular to the plane of the lamina. This coordinate system is shown in Figure 5.
In an orthotropic material, linear elasticity is defined by giving the “engineering constants”, specifically:

- the three moduli \( E_1, E_2, E_3 \);
- the three Poisson’s ratios \( \nu_{12}, \nu_{13}, \nu_{23} \);
- the three shear moduli \( G_{12}, G_{13}, G_{23} \).

In order to determine the engineering parameters \( E_1, E_2, E_3, \nu_{12}, \nu_{13}, \nu_{23}, G_{12}, G_{13}, G_{23} \) of a fiber-reinforced composite material, the micromechanics theoretical approach is used [9]. It can also be determined experimentally using an appropriate test specimen made up for the material. All these constants are associated with the local coordinate system. If a material is orthotropic, the strain-stress relations are written (in that system) in the following form [8],

\[
\begin{bmatrix}
\varepsilon_{11} \\
\varepsilon_{22} \\
\varepsilon_{33} \\
\gamma_{12} \\
\gamma_{13} \\
\gamma_{23}
\end{bmatrix}
= \begin{bmatrix}
1/E_1 & -\nu_{21}/E_2 & -\nu_{31}/E_3 & 0 & 0 & 0 \\
-\nu_{12}/E_1 & 1/E_2 & -\nu_{32}/E_3 & 0 & 0 & 0 \\
-\nu_{13}/E_1 & -\nu_{23}/E_2 & 1/E_3 & 0 & 0 & 0 \\
0 & 0 & 0 & 1/G_{12} & 0 & 0 \\
0 & 0 & 0 & 0 & 1/G_{13} & 0 \\
0 & 0 & 0 & 0 & 0 & 1/G_{23}
\end{bmatrix}
\begin{bmatrix}
\sigma_{11} \\
\sigma_{22} \\
\sigma_{33} \\
\tau_{12} \\
\tau_{13} \\
\tau_{23}
\end{bmatrix}
\]  
(2-5)

Transverse isotropy is a special subclass of orthotropy, which is characterized by a plane of isotropy at every point. If the 2-3 plane is considered to be that plane, transverse isotropy requires that \( E_1 = E_∥ \), \( E_2 = E_3 = E_⊥ \), \( \nu_{12} = \nu_{13} = \nu_{∥∥} \), \( \nu_{21} = \nu_{31} = \nu_{⊥⊥} \) and \( G_{12} = G_{13} = G_{∥∥} \) (\( \parallel \) stands for the parallel direction to the fibers and \( \perp \) for the transverse direction). From now on, subscript 1 will refer to the direction parallel to the fibers (\( \parallel \)) and subscript 2 to the transverse direction (\( \perp \)).

For a transversely isotropic material, there are only five independent properties, as \( G_{∥∥} = \frac{E_{∥∥}}{2(1+\nu_{∥∥})} \) [10] and \( \frac{E_1}{\nu_{12}} = \frac{E_2}{\nu_{21}} \), as a result of the application of the Maxwell-Betti reciprocal relations as described in [11]. Summing up, the five properties are \( E_1, E_2, \nu_{12} \) and \( G_{12} \).

Equation (2-5) was derived for a full three-dimensional case. The main components of a composite fuselage panel (skin and stringers) consist of multiple single unidirectional (UD) layers that are relatively thin, so the simplified condition of plane stress is accurate, and loading can be considered to be in the plane of the layer. Under plane stress conditions, only the values of \( E_1, E_2, \nu_{12}, G_{12} \) are required to define an orthotropic material, because the variables \( \nu_{13}, \tau_{13} \) and \( \tau_{23} \) are considered to be zero, which means that no through-thickness stresses are introduced for the thin layer. The shear moduli \( G_{13} \) and \( G_{23} \) are included to shell elements because they may be required for modelling...
transverse shear deformation in a shell [4]. Thus, the in-plane stress-strain relations are simplified to [6]:

\[
\begin{pmatrix}
\varepsilon_1 \\
\varepsilon_2 \\
\gamma_{12}
\end{pmatrix} =
\begin{bmatrix}
\frac{1}{E_1} & -\frac{\nu_{12}}{E_1} & 0 \\
\frac{\nu_{12}}{E_1} & \frac{1}{E_2} & 0 \\
0 & 0 & \frac{1}{G_{12}}
\end{bmatrix}
\begin{pmatrix}
\sigma_1 \\
\sigma_2 \\
\tau_{12}
\end{pmatrix}
\]  

(2-6)

which can be written in the form

\[
\{\varepsilon\} = [S]\cdot\{\sigma\}
\]  

(2-7)

where \([S]\) is called the reduced compliance matrix.

The \(S_{12}\) entry is accounted for the fact that the Poisson’s ratio \(\nu_{21}\) is implicitly given as \(\nu_{21} = \frac{\nu_{21}}{E_1}\). It is important to understand that, although in plane stress there can be no shear strains in the 1-3 and 2-3 planes (\(\gamma_{13} = \gamma_{23} = 0\)), the strain in the 3-direction is not zero. A tensile normal stress \(\sigma_1\) causes not only extension of the element in the 1-direction as well as contraction in the 2- and 3-direction, therefore \(\varepsilon_3 \neq 0\) [8].

Similarly, the stress-strain equations are given below.

\[
\{\sigma\} = [Q]\cdot\{\varepsilon\}
\]  

(2-8)

in which \([Q]\) is the so-called stiffness matrix.

\[
\begin{pmatrix}
\sigma_1 \\
\sigma_2 \\
\tau_{12}
\end{pmatrix} =
\begin{bmatrix}
\frac{E_1}{1-\nu_{21}\cdot\nu_{12}} & \frac{E_2\cdot\nu_{12}}{1-\nu_{21}\cdot\nu_{12}} & 0 \\
\frac{E_2}{1-\nu_{21}\cdot\nu_{12}} & \frac{E_2}{1-\nu_{21}\cdot\nu_{12}} & 0 \\
0 & 0 & \frac{1}{G_{12}}
\end{bmatrix}
\begin{pmatrix}
\varepsilon_1 \\
\varepsilon_2 \\
\gamma_{12}
\end{pmatrix}
\]  

(2-9)

The stiffness and strength of fiber-reinforced composites come from the fibers, which means that the material is stiffer and more resistant in the 1-direction than in the transverse (2 and 3) directions.

A fibre-reinforced composite is made of a finite number of single layers, with different fibre orientations with respect to the global coordinate system. In the assembly of the laminate, the constitutive relation of each ply is transformed to that global system, and as a result a new global stiffness matrix is formed, so that the entire laminate is represented by a single constitutive relation.

Figure 6 shows the relation between local and global coordinate systems. The first one is represented with its 1-axis parallel to the fiber direction and the global one is rotated by an angle of \(\theta\) around the z-axis. The constitutive relations must be transformed into the global coordinate system in order to calculate the stiffness of the entire composite. The necessary transformations can be found in Reddy [9].
The CLPT is an extension of the Classical Plate Theory to laminated plates. In this theory, the in-plane displacements are assumed to vary linearly through the thickness and the transverse displacement is assumed to be constant through the thickness, which means that there is no strain in the thickness direction. This underlying two-dimensional assumption (2D) is accurate as long as the thickness of the laminate is small (at least two orders of magnitude less than the in-plane dimensions) [6].

Figure 7 gives the definition of lever arms \(z_k\) of the single layers of the laminate, according to the CLPT theory, which are required to calculate the stress and bending resultants.

\[
\begin{align*}
n_x &= \int_{z_0}^{z_n} \sigma_x \, dz \\
n_y &= \int_{z_0}^{z_n} \sigma_y \, dz \\
n_{xy} &= \int_{z_0}^{z_n} \tau_{xy} \, dz \\
m_x &= \int_{z_0}^{z_n} \sigma_x \cdot z \, dz \\
m_y &= \int_{z_0}^{z_n} \sigma_y \cdot z \, dz \\
m_{xy} &= \int_{z_0}^{z_n} \tau_{xy} \cdot z \, dz
\end{align*}
\]

The combined matrix that relates the six stress and bending resultants with the six reference deformation components is called the laminate stiffness matrix. It comprises an extensional or membrane stiffness matrix, a flexural or bending stiffness matrix and a coupling stiffness matrix. For a more detailed description of the CLPT theory, the reader is referred to Reddy [9].
2.3. Damage mechanisms and models

For an efficient design of composite structures, the damage of composite materials (failure mechanisms) must be considered, so modeling the material damage and failure is a key task. This section will describe the damage mechanisms and models, as well as the various approaches and numerical procedures relevant to characterize the onset and propagation of damage in stiffened composite structures.

2.3.1. Damage mechanisms

In composite materials, the extreme anisotropy in both stiffness and strength properties and the presence of two different constituents (fibers and matrix) result in various failure/damage mechanisms at distinct levels [12]. Those mechanisms that are relevant to stiffened composite structures can be divided in intra-laminar damage (ply failure), inter-laminar damage (delamination) and a typical failure in stiffened structures known as skin-stringer debonding, as follows [13]:

Damage Mechanism

Intra-laminar

Fiber damage

Fiber tension (fiber rupture)

Fiber compression (fiber kinking)

Matrix damage

Matrix tension (matrix cracking)

Matrix compression (matrix crushing)

Inter-laminar

Delamination

Skin-Stringer debonding

Figure 8: Damage mechanisms in composite stiffened structures

The development of intra-laminar damage mechanisms is critical to the collapse of composite structures. These failure modes are illustrated in Appendix A. Fiber failure is one of the simplest failure mechanisms to identify. It is also the principal damage mechanism causing collapse, as in FRP composites the fibers resist the majority of the applied loads [14]. This type of damage can be caused by breakage of the fibers in tension or by fiber micro-buckling (kinking) in compression. The onset of fiber failure typically leads to a significant loss of load-carrying capacity and is taken as the point of final structural collapse, as demonstrated by numerous investigations conducted by Orifici [5], Thomson et al. [14], Caputo et al. [15], and others.

Matrix failure is another ply damage mechanism which is also important to capture. This damage mechanism can be divided in matrix cracking in tension and matrix crushing in compression. The work
of Orifici in [5] and Degenhardt et al. in [2] have shown that this type of failure results in local ply softening and, when associated with buckling, can lead to reduction of the buckling load.

Fiber-matrix shear acts also as an individual failure mode, but it doesn’t have much influence on the global behavior of the structures.

Inter-laminar damage is one of the critical damage mechanisms for laminated composite materials and can result in significant structural degradation, so it is crucial for the study of postbuckling composite structures. Delamination is the most usual form of inter-laminar damage and occurs due to high through-thickness stresses overcoming the inter-laminar bond strength between two plies [5]. This type of failure consists in the separation between internal layers of a composite laminate (as illustrated likewise in Appendix A) and can occur as a result of impact loading, for instance. Delamination leads to a significant reduction in the compressive load-carrying capacity of a composite structure [12].

For composite aerospace structures, the skin and stiffeners are either co-cured as a complete laminate or manufactured separately and adhesively bonded [14]. In the last case, besides delamination between plies of the composite, the most severe damage mechanism is skin-stringer debonding. This type of damage involves detachment of the stringers from the skin in stiffened panels, which can result in an “explosive” form of failure.

2.3.2. Damage characterization and modelling

There are two different approaches to characterize the onset and growth of damage in composite structures:

1) **Continuum damage mechanics:**

Within the framework of Continuum Damage Mechanics (CDM), maximum allowable strength-based criteria are commonly used to predict failure events in composite structures and are defined by specifying maximum (allowable) strengths for a material [5]. This is explained with further detail in the next section.

2) **Fracture mechanics:**

Classical fracture mechanics is a theory based on the growth of existing defects/cracks in the structure. According to Gliszczynski and Tobiak [16], the application of fracture mechanics to composite materials is more complex compared to its application to isotropic materials. The heterogeneity and anisotropy of composite materials causes that the orientation of the crack fronts depends not only on the load, geometry and boundary conditions, but also on the morphology of the material. In fracture mechanics theory, crack propagation is predicted by comparing the computed values of the stress intensity factors, or the components of the strain energy release rate, with the corresponding critical values, taken as material properties [12]. The crack propagation can be split, with respect to the released strain energy, in three different modes: mode I (opening/peeling), mode II (sliding) and mode III (tearing), as illustrated in Figure 9.
Delamination is usually predicted using fracture mechanics. To date, FEA using this theory has been limited “due to the complexities involved in monitoring crack progression and a typical requirement for a fine mesh around the crack front, which usually requires either a highly dense mesh or computationally expensive re-meshing” [5]. Following the work presented by Krueger et al. [17], Linear Elastic Fracture Mechanics (LEFM) was proven useful for characterizing the onset and growth of delamination in composite laminates. On the other hand, an approach using fracture mechanics has recently been added to the commercial FE code Abaqus/Standard to model crack progression, which is called the Virtual Crack Closure Technique (VCCT). Moreover, cohesive elements can be used for modelling delamination in composite materials. These methods will be described in the next section.

The implementation of all types of damage mechanisms in the FE models has been proved to be a difficult and time-consuming task. This section will give a review on some of the numerical methods that are commonly used for representing damage in composite stiffened structures in Abaqus FE models. Those methods comprise Hashin’s criteria, cohesive elements and the eXtended FE Method (XFEM).

2.3.2.1 Hashin criteria

Continuum Damage Mechanics is the modelling approach used in this work to model intra-laminar failure of the CFRP parts (skin and stiffeners). In CDM, the failure modes are represented and modelled by the degradation (reduction) of the material stiffness to implement the loss in load-carrying capacity [16]. Strength-based failure criteria are used in CDM to predict the onset of failure and the progression of damage is achieved by introducing damage variables into the material constitutive law.

Hashin damage initiation criteria

In Abaqus, the ply failure is predicted with the implementation of Hashin and Rotem damage initiation criteria [18] and Hashin’s failure criteria for unidirectional fiber-reinforced composites [19]. These criteria can only be used in Abaqus with elements with a plane stress formulation like shell, continuum shell and membrane elements. It considers 4 different damage initiation modes as follows [4]:

- **Fiber Tension (breakage):**

  \[
  F_f^t = \left( \frac{\sigma_{11}}{X_T} \right)^2 + \alpha \left( \frac{\sigma_{12}}{S_L} \right)^2
  \]  

  \(2-11\)
• Fiber Compression (kinking):

\[ F^c_f = \left( \frac{\sigma_{11}}{X_C} \right)^2 \]

(2-12)

• Matrix Tension (cracking):

\[ F^t_m = \left( \frac{\sigma_{22}}{Y_C} \right)^2 + \left( \frac{\sigma_{12}}{S_L} \right)^2 \]

(2-13)

• Matrix Compression (crushing):

\[ F^c_m = \left( \frac{\sigma_{22}}{2S_T} \right)^2 + \left[ \left( \frac{Y_C}{2S_T} \right)^2 - 1 \right] \frac{\sigma_{22}}{Y_C} + \left( \frac{\sigma_{12}}{S_L} \right)^2 \]

(2-14)

In the above equations (i) \( X_T \) and \( X_C \) are the tensile and compressive strengths, respectively, in the fiber direction, (ii) \( Y_T \) and \( Y_C \) are the tensile and compressive strengths, respectively, in the transverse direction, (iii) \( S_L \) and \( S_T \) denote, correspondingly, the longitudinal and transverse shear strengths, and (iv) \( \alpha \) is a coefficient that determines the contribution of the shear stress to the fiber tensile initiation criterion. To obtain the model proposed by Hashin and Rotem [18] \( \alpha = 0 \) and \( S_T = Y_C/2 \) and for the model proposed by Hashin [19] \( \alpha = 1 \) is adopted.

Furthermore, \( F^t_f, F^c_f, F^t_m \) and \( F^c_m \) are indexes that indicate whether a damage initiation criterion has been satisfied or not. When any of the indexes exceeds 1.0 it means that the initiation criterion has been met and thus damage has begun.

\( \sigma_{11}, \sigma_{22} \) and \( \sigma_{12} \) are components of the effective stress tensor which is computed from [4] :

\[ \tilde{\sigma} = M \sigma \]

(2-15)

where \( \sigma \) is the apparent/true stress and \( M \) the damage operator matrix. Abaqus uses the model proposed by Matzenmiller et al. [20] to compute the tensor \( M \) as follows:

\[
M = \begin{bmatrix}
1 & 0 & 0 \\
\frac{1}{(1 - d_f)} & 0 & 0 \\
0 & \frac{1}{(1 - d_m)} & 0 \\
0 & 0 & \frac{1}{(1 - d_s)}
\end{bmatrix}
\]

(2-16)

\( d_f, d_m \) and \( d_s \) are internal variables that characterize fiber, matrix and shear damage, respectively. These three variables are derived from the four damage variables \( d^t_f, d^c_f, d^t_m \) and \( d^c_m \) that characterize fiber and matrix damage in tension and compression, corresponding to the four damage mechanisms previously discussed, as follows,
\[ d_f = \begin{cases} d_f^t & \text{if } \sigma_{11}^f > 0 \\ d_f^c & \text{if } \sigma_{11}^f < 0 \end{cases} \]

\[ d_m = \begin{cases} d_m^t & \text{if } \sigma_{22}^m > 0 \\ d_m^c & \text{if } \sigma_{22}^m < 0 \end{cases} \]

\[ d_s = 1 - (1 - d_f^t)(1 - d_f^c)(1 - d_m^t)(1 - d_m^c) \]  \hspace{1cm} (2-17)

Prior to any damage initiation the material is linear elastic and the damage operator matrix is equal to the identity matrix, so \( \sigma = \sigma \). Once damage initiation and evolution has occurred for at least one mode, the damage operator \( M \) becomes relevant in the criteria for damage initiation of other modes.

In Abaqus, the damage initiation output variables associated with each initiation criterion (indexes \( F_f^t \), \( F_f^c \), \( F_m^t \) and \( F_m^c \)) are shown in Appendix A.

**Damage evolution law**

The damage evolution for fiber-reinforced composite materials in ABAQUS [4]:

- “Requires Linear Elastic behavior to the undamaged material”;
- “Assumes that damage is characterized by progressive degradation of material stiffness, leading to material failure”;
- “Is based on energy dissipation during the damage process”;
- “Includes the removal of elements from the mesh (optional)”.

Once any of the damage initiation criteria (equations 2-11 to 2-14) is satisfied, the effect of damage is taken into account by reducing the values of stiffness coefficients and the response of the material is computed from

\[ \sigma = C_d \varepsilon \]  \hspace{1cm} (2-18)

where \( \varepsilon \) represents the strain and \( C_d \) the damage elasticity matrix, which has the following form [4],

\[
C_d = \frac{1}{\phi} \begin{bmatrix}
(1 - d_f^t)E_1 & (1 - d_f^t)(1 - d_m^t)\nu_{12}E_2 & 0 \\
(1 - d_f^t)(1 - d_m^t)\nu_{12}E_2 & (1 - d_m^t)E_2 & 0 \\
0 & 0 & (1 - d_s)\mu \phi
\end{bmatrix}
\]  \hspace{1cm} (2-19)

where \( \phi = 1 - (1 - d_f^t)(1 - d_m^t)\nu_{12} \), \( d_f^t, d_m \) and \( d_s \) reflect the current state of fiber, matrix and shear damage, respectively.

When the material exhibits strain-softening behavior, the constitutive model expressed in terms of strain-stress equations results in strong mesh dependency. To alleviate that, a characteristic length is introduced into the formulation. For shells the characteristic length \( L_c \) is computed as the square root of the area of the reference of the element [21]. Using the characteristic length, the stress-strain constitutive model is transformed into the stress-displacement relation as shown in Figure 10.
The positive slope of the curve corresponds to linear elastic material behavior, whereas the negative slope is achieved after damage initiation and evolution of the respective damage variables. The evolution of each damage variable is governed by an equivalent displacement $\delta_{eq}$. Hence, each damage mode is represented as a 1D stress-displacement problem even though the stress and strain fields of the problem are 3D [21]. The equivalent displacement for each of the four damage modes are defined and explained in greater detail in [4]. The damage variable for a particular mode is given by the following expression:

$$\begin{align*}
d &= \frac{\delta^f_{eq} (\delta_{eq} - \delta^0_{eq})}{\delta_{eq} (\delta^f_{eq} - \delta^0_{eq})} \\
\text{(2-20)}
\end{align*}$$

where $\delta^0_{eq}$ is the initial equivalent displacement at which the initiation criterion for a given mode was met and $\delta^f_{eq}$ is the maximum value of displacement at which the material is completely damaged.

The evolution of the damage variables as well as the value of $\delta^0_{eq}$ depend on the elastic stiffness and the strength parameters. The critical energy release rate, $G^c$, also known as fracture energy, must be specified for each failure mode, which corresponds to the area of the triangle of Figure 10. Therefore, in addition to the stiffness and strength, four values of critical $G^c$ must be provided.

**Maximum degradation, element removal and viscous regularization**

In Abaqus [4], by default, the upper bound to all damage variables is $d_{max} = 1.0$. By allowing element removal, an element is deleted once damage variables for all failure modes at all materials points reach $d_{max}$. If an element is removed, the output variable STATUS is set to zero for the element and it offers no resistance to further deformation [4].

Applying a load to a node that is not attached to an active element will cause convergence difficulties. On the other hand, the material softening behavior and stiffness degradation often lead to convergence problems in implicit analysis programs such as Abaqus/Standard. To overcome this issue, Abaqus allows the implementation of the viscous regularization scheme, at which a damage variable is defined by the evolution equation [4],

$$\dot{d}_v = \frac{1}{\eta} (d - d_v)$$

(2-21)
where \( \eta \) is the viscosity coefficient and \( d \) is the damage variable in the inviscid model. The damage elasticity matrix, \( C_d \), is computed using the viscous values of damage variables (\( d_v \) instead of \( d \)) for each failure mode.

In Abaqus, the output variables related specifically to damage evolution in fiber-reinforced composites are given in Appendix A.

### 2.3.2.2 The eXtended FE Method and Virtual Crack Closure Technique

The eXtended Finite Element Method (XFEM) is an extension of the conventional FEM. It is a numerical technique for describing and tracking the motion of a crack and it allows [22]:

- The entire Crack to be represented independently of the mesh, and so remeshing is not necessary to model crack growth;
- The presence of discontinuities in an element by enriching degrees of freedom (DOF) with special displacement functions.

The XFEM in Abaqus describes shape, position and direction of the crack and extends the degree of freedom of elements leading to a new enriched displacement vector:

\[
u = \sum_i N_i(x) \left[ u_i + H(x) a_i + \sum_{\alpha=1}^{4} F_\alpha(x) b_i^\alpha \right]
\]

where \( N_i \), \( u_i \), \( H \), \( a_i \), \( F_\alpha \) and \( b_i^\alpha \) denote the nodal shape functions, the nodal displacements, the discontinuous jump function, the nodal enriched degrees of freedom at the whole crack, the asymptotic crack-tip function and the nodal enriched degrees of freedom at the crack tip, respectively.

For a more thorough description of those new functions, as well as other relevant aspects related to the XFEM method, the reader is referred to [22].

The Linear Elastic Fracture Mechanics (LEFM) can be used within the XFEM framework by means of the Virtual Crack Closure Technique (VCCT), described next. This approach is based on the calculation of the strain energy release rate at the crack tip. The advantage of using XFEM with VCCT is that this approach can be used when no initial crack is present. Therefore, damage initiation must be specified in the material property definition, VCCT becomes active when damage initiation criteria are met, and a crack propagates according to XFEM.

The VCCT represents a highly successful technique that has been used by many authors in research and also in industry to predict crack growth in composite structures. Lauterback et al. [23] and Krueger et al. [17] have successfully applied the VCCT method in order to predict delamination and skin-stiffener debonding in composite panels. In this work, VCCT was used within XFEM, as stated previously, to calculate the strain energy release rates at the crack tip.
The VCCT theory is based on two assumptions [24]:

1) The energy released in crack growth is equal to work required to close the crack to its original length;
2) The crack growth does not significantly modify the state at the crack tip.

Assuming that the crack closure is governed by linear elastic behavior, the energy to close the crack (or to open it) is calculated, for a 2D problem, from the following equations,

\[ G_I = -\frac{1}{2} F_j \Delta U_i \Delta A \]  \hspace{2cm} (2-23)

\[ \Delta A = \delta a b \]  \hspace{2cm} (2-24)

where \( F_j, \Delta U_i, \delta a, b \) and \( G_I \) denote, respectively, the reaction force at node \( j \) (Figure 11), the displacement between released nodes at \( i \), the length of the element at the crack front, the width and the energy release rate. The nodes \( i \) and \( i' \) will start to release when the following criterion is met,

\[ G_I \geq G_I^c \]  \hspace{2cm} (2-25)

where \( G_I^c \) is the mode I fracture toughness.

Figure 11: VCCT method for pure Mode I [24]

The VCCT method can be applied to a 3D model. The application of this method for 8-node solid elements is depicted in Appendix-A. According to Camanho [12], the VCCT method can be computationally effective when all the elements at the crack tip have the same dimensions in the crack growing dimension and when sufficiently refined meshes are used.
2.3.2.3 Cohesive FE models

Cohesive FE models can be implemented in both FE packages, Abaqus/Standard and Abaqus/Explicit [4]. These interface elements combine the strength of materials formulation (damage mechanics) for crack initiation with fracture mechanics for crack propagation [12], and are increasingly being used by researchers to model adhesive failure, delaminations and debonds in composite structures.

The cohesive element formulation allows the combination of several damage mechanisms acting simultaneously on the same material [4]. Modelling with cohesive elements has many important advantages over other approaches, especially for delamination and debonding, since they have the capacity to predict both initiation and growth of damage in the same analysis, as well as to incorporate both strength and fracture mechanics damage theories [5]. However, a fine mesh is required for the analysis to remain accurate, so the application of cohesive elements to large structures can become problematic. Moreover, it cannot differentiate between mode II and III crack opening mode and the exact location of the crack front can be difficult to define.

The cohesive behavior is defined by a traction-separation law (TSL), which assumes linear elastic behavior followed by the initiation and evolution of damage [4]. This law uses the bond separation distance instead of physical strain as independent axis [25]. Figure 12 defines the relationship between the gap opening ($\delta$) and traction ($\tau$) across the cohesive interface.

The bond material is assumed to behave with zero ductility until it fails, which means that the initial response of the cohesive element is assumed to be linear. Once damage initiation criterion is met, i.e., after the element passes the strength limit ($\tau^0$) of the bond material, the stiffness is gradually reduced. The loss of stiffness of the interface continues until it reaches a value of zero, at which point the substructures are completely delaminated ($\delta^F$). When the bond material starts to fail, it releases a finite amount of energy per unit growth of the crack [25]. The work done in reducing the material stiffness to zero is equal to the fracture toughness, also known as the critical energy release rate ($G_c$). Therefore, the fracture energy is equal to the area under the traction-separation curve.

The damage initiation is predicted using certain criteria that must be specified by the user. Several damage initiation criteria are available in Abaqus, ones based on maximum allowable stresses and

\[ \delta^F = \frac{1}{K} \left( G_c \right) \]

\[ d = \frac{\delta}{\delta^F} \]

\[ d = 0 \]

\[ 0 < d < 1 \]

\[ d = 1 \]

\[ \tau^0 \]

\[ \delta^0 \]

\[ \delta^F \]

\[ \tau \]

\[ G_c \]

\[ (1-d)K \]

\[ \delta \]

Figure 12: Traction-separation cohesive behaviour
others based on maximum allowable strains. Amongst the existing ones, those based on the maximum allowable stresses which allow modelling the onset of damage considering mode I, II and III contributions are described next: the maximum nominal stress criterion (MAXS) and the quadratic nominal stress criterion (QUADS).

The maximum nominal stress criterion (MAXS) implies that damage in the bond layer starts when the maximum nominal stress ratio reaches a value of 1, according to the following expression [4],

$$\max \left\{ \frac{\sigma_{nn}}{S_{nn}}, \frac{\sigma_{s1}}{S_{s1}}, \frac{\sigma_{s2}}{S_{s2}} \right\} = 1 $$ \hspace{1cm} (2-26)

where $\sigma_{nn}$ represent the normal stress; $\sigma_{s1}$ and $\sigma_{s2}$ the two perpendicular shear tractions; $S_{nn}$, $S_{s1}$ and $S_{s2}$ denote the peak (maximum) values of the nominal stress in the cohesive layer.

The quadratic nominal stress criterion (QUADS) infers that damage in the cohesive layer is initiated when subject to a quadratic stress criterion of the form,

$$\left( \frac{\sigma_{nn}}{S_{nn}} \right)^2 + \left( \frac{\sigma_{s1}}{S_{s1}} \right)^2 + \left( \frac{\sigma_{s2}}{S_{s2}} \right)^2 = 1 $$ \hspace{1cm} (2-27)

The damage propagation is fundamentally based on energy principles and describes the rate at which the material stiffness is degraded once damage starts [4]. The adhesive layer has a contribution of mode I and II in the failure process, i.e., it is most likely mix loaded, so in order to define the dependence of the fracture energies on the mode mix, two criteria can be generally used [4],

- Power Law form;
- Benzegagh-Kenane (BK) criterion (used in this work).

The BK criterion was suggested by Benzegagh and Kenane [26]. This criterion is particularly useful when the critical fracture energies during deformation purely along the first and second shear directions are the same, i.e., $G_{IIC} = G_{IIC}$ [4]. The mix-mode fracture energy is given by:

$$G^c = G_T + (G_{II} - G_T) \left( \frac{G_S}{G_T + G_S} \right)^\eta $$ \hspace{1cm} (2-28)

where $G_S = G_{II} + G_{III} = 2G_{II}$.

Fracture (total separation) is expected when the total energy release rate, $G_T$, exceeds $G^c$.

In the same way as explained in section 2.3.2.1 for damage in FRP composites, an option for element removal is available in Abaqus. Using cohesive elements, the element deletion (Figure 13) is often appropriate for modeling separation of components and complete fracture.
On the other hand, Abaqus also allows the implementation of the viscous regularization scheme to the cohesive zone model, which is useful to overcome convergence difficulties resulting from material softening and stiffness degradation.

The output variables related to cohesive elements with traction-separation response are given in Appendix A.

2.4. Buckling, postbuckling and strength of stiffened panels

There are several studies in the literature on the buckling and strength of curved panels. In order to avoid a lengthy and cumbersome description of all these works, we opted to focus our attention to the results of the project COCOMAT. Thus, this section presents a brief overview of the main outcomes of the COCOMAT project. In this project, improved FEA tools were developed to investigate the postbuckling behavior of composite structures under axial compression. In order to validate the numerical results, the COCOMAT project researchers created new experimental data bases for curved stringer-stiffened CFRP panels, since appropriate experimental tests were not available at the beginning of the project.

Under compression, composite stiffened CFRP panels undergo buckling, degradation and final collapse. Figure 14 illustrates an experimental load-shortening curve of an axially compressed stiffened panel, in which 3 remarkable load levels can be distinguished. The lowest one, a local buckling region, where buckling waves develop in the skin between the stiffeners, occurs in this structures as the first buckling mode. Afterwards, a slight stiffness reduction occurs. The second level is the onset of buckling of the stiffeners and is represented by a higher reduction of the axial stiffness. Collapse is the highest level and is specified by the point of the curve where a sharp decrease in the axial stiffness occurs [2]. The shortening between the first local buckling load and collapse is called the postbuckling shortening. Additional buckling mode shapes may develop as these structures have the tendency to switch (“snap”) to high-order buckling shapes [5].
Concerning the experimental tests, an undamaged 5-blade stiffened curved panel was manufactured and tested at the DLR (German Aerospace Institute). The panel's dimensions are shown in Appendix A. Skin and stringers were manufactured separately and bonded using an adhesive [8]. The top and bottom edge of the panel were encased in resin to ensure a homogeneous distribution of the applied displacement. The tested panel was similar to P23 panel of the POSICOSS project, but no restraints were applied on the lateral edges. The tested panel, named Design 1 (D1), was placed in the testing apparatus of the DLR and the measuring equipment was initially calibrated. A quasi-static monotonically increasing displacement of 4 mm was applied in the axial direction. The testing apparatus and the panel after collapse can be seen also in Appendix A. The experimental data was obtained using strain gages and by means of ARAMIS, which is a 3D optical deformation measurement system based on photogrammetry [2].

The load-shortening curve obtained from the experimental test is given in Figure 15. The global buckling of the panel occurred for an applied axial shortening of about 0.97 mm [8], where a reduction in the load occurs and the axial stiffness displays a high decrease. At an axial shortening of about 1.72 mm, another global buckling shape is developed, and finally the collapse of the panel took place at a shortening of 2.71 mm and at a load of 83.6 KN.

![Figure 15: Load-shortening curve of the experimental test [4]](image)
Figure 16 depicts the deformation pattern measured experimentally (buckling and postbuckling mode shapes) of the COCOMAT panel at different values of axial shortening. These mode shapes will be further discussed in Chapter 5 for purposes of comparison with the ones obtained numerically.

The numerical simulations of COCOMAT were performed employing geometrical nonlinear analysis with explicit and implicit solution procedures. The material behavior was considered linear elastic up to the maximum allowable stresses. The onset of damage of the composite plies was predicted with Tsai-Wu and Tsai-Hill failure criteria. However, the principal failure mechanism was implemented in the adhesive layer connecting the skin and the stringers [8], because skin-stringer separation was measured and considered to be the most severe form of failure. In order to model this type of damage, the partners of DLR developed user subroutines programmed in FORTRAN to be used with Abaqus, which considered the skin-stringer debonding using stress-based failure criteria [2].

The load-shortening curve of the numerical analysis with Abaqus without any type of damage included is given in Figure A-9 (Appendix A). Figure 17 shows the load-shortening curve of different versions of the USDFLD, the subroutine mostly used within the COCOMAT project, which simulates the damage of the adhesive layer by decreasing the Young’s modulus (E) to a small fraction of the initial value for the FEs in which the maximum allowable stress was reached [2].
Comparing the results, one can see that the version of the USDFLD with a reduction of the Young’s modulus to 0.1% when failure was detected gave a very close prediction of the experimental load-carrying capacity, showing a good agreement except for the shortening ($u = 2.71$ mm) at which the structure collapsed [2]. However, this good agreement in terms of load-shortening curves was not as good when comparison was made between areas of damaged adhesive. The degree of damage predicted by the numerical simulation was that the adhesive layer was almost completely damaged, as illustrated in figure 18. However, during the experimental testing, failure of the adhesive was not observed throughout since the skin-stringer debonding was only visible in two regions after the panel had collapsed (as depicted in Figure A-10). Furthermore, the numerical and experimental global deformation patterns are different for all versions of the USDFLD, as the asymmetry seen in the experiment wasn’t detected. This fact is illustrated in Figure 19, for the point of collapse.

![Figure 18: Failure propagation of the adhesive layer at 4 load levels: A-70 kN; B-75 kN; C-79 KN; D-83 kN](image1)

![Figure 19: Global deformation pattern at collapse for different versions of the USDFLD subroutine. The red/yellow colours represent outward displacement and the blue/green inward displacement (adapted from [8])](image2)

Hence, the simulated model did not give a realistic prediction of failure. This was also true for the additional versions of the USDFLD. Therefore, further improved damage models were needed at the end of the COCOMAT project.
3. FE Model

The FE models developed for the purpose of this thesis consisted of seven different curved stringer-stiffened panels, which were assumed to be representative of a fuselage section. Each panel comprised a skin with cylindrical shape and longitudinal stiffeners (stringers), created as separate parts, and adhesively bonded. The seven panels were made of carbon/epoxy IM7/8552 prepreg tape and differed from each other either by the number of stringers or by the stringer section geometry considered.

All numerical models were created using conventional shell elements, in which a laminate material definition according to CLPT was applied. The COCOMAT panel (D1) was taken as start design for the purpose of validation of the experimental data (available within the COCOMAT project), as well as for comparison with the results obtained numerically by Degenhardt et al. [2], previously described in Chapter 2. Different modeling approaches for material damage were conducted in the FEA, including damage of the composite material and damage of the adhesive. Furthermore, the postbuckling analysis of panels in compression was extended to several panel designs and an additional analysis of all seven panels subjected to bending was also developed. Nonlinear analyses up to collapse were performed using an incremental Newton-Raphson method.

The FE analysis procedure for this type of structures was summarized in section 2.1 and is depicted in Figure 20. It consists of four stages: the preprocessing, a linear eigenvalue analysis to incorporate imperfections into the model, a nonlinear analysis and finally, the postprocessing.

![Figure 20: Analysis procedure in Abaqus](image)
3.1. Geometry

Several panels were investigated. The reference panel is labelled as T5 panel because it has 5 T-shaped stingers. Then, a variation of the number of stringers was performed to check their influence: panels T4 and T6 containing 4 and 6 T-shaped stringers were also modelled. Then, in order to study the influence of stringer geometry, other four different shapes (I, C, J and Ω) of stringer were considered (each panel always with 5 stringers).

The geometry of the T5 panel was based on the COCOMAT panel D1, manufactured by Aernnova Engineering Solutions and tested by the Institute of Composite Structures and Adaptive Systems of DLR (German Aerospace Center). It consists of a thin CFRP skin stiffened with five CFRP stringers. The geometric data of the panel is given in Table 1.

<table>
<thead>
<tr>
<th>Panel length</th>
<th>L = 780 mm</th>
<th>Stringer blade height</th>
<th>h = 14 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free length</td>
<td>Lf = 660 mm</td>
<td>Stringer width</td>
<td>b = 32 mm</td>
</tr>
<tr>
<td>Radius</td>
<td>r = 1000 mm</td>
<td>Ply thickness of all CFRP layers</td>
<td>t = 0.125 mm</td>
</tr>
<tr>
<td>Arc length</td>
<td>a = 560 mm</td>
<td>Laminate set up of the skin</td>
<td>[90, +45, -45, 0]s</td>
</tr>
<tr>
<td>Number of stringers</td>
<td>5</td>
<td>Laminate set up of the stinger flange</td>
<td>[(45, -45)3, 06]</td>
</tr>
<tr>
<td>Distance between stringers</td>
<td>d = 130 mm</td>
<td>Laminate set up of the stinger blade</td>
<td>[(45, -45)3, 06]s</td>
</tr>
</tbody>
</table>

Figure 21 illustrates the geometry and dimensions of the T-shaped stringers. The skin thickness direction is made of 8 composite plies, while the stringer flange and blade are made of 12 and 24 plies, respectively. Each ply has a uniform thickness of t = 0.125 mm, which means that the skin has a total thickness of t = 1.0 mm, while the stringer flange and blade have a thickness of t = 1.5 mm and t = 3 mm, respectively. Only uni-directional (UD) layers were used to model the stringers.

![Figure 21: Geometry and dimensions of the t-shaped stringers used in the COCOMAT project [8]](image-url)
In reality, the layup of the stringer flange \( [(45, -45)_2, 0_4] \) is split in the center and bended 90° to form the stringer blade. However, in the FE model developed, the stingers were represented as separate shell parts (flange and blade), as depicted in Figure 22, modeled by conventional 2D shell elements and connected by a row of nodes.

![Figure 22: Midsurface shell approach for the T-shaped stringers](image)

The adhesive layer between the skin and stringers was modeled with 3D FEAs as described in greater detail in section 3.2. The thickness of the adhesive was set to be equal to \( t = 0.2 \) mm as this value was the one used in the experimental test.

The additional six panels considered in this work were designed as follows. Panels T4 and T6 present identical T-shaped stringers but differ in their number, as the first contains four and the latter contains six stringers. These panels were created to evaluate the influence of the number of stringers on the structural behavior of the stiffened panels.

The remaining panel designs comprised the same number of stringers as the original one (panel T5), but the stringer geometry was varied. Table 2 shows the dimensions and geometry of those panels. The stringer blade height of approximately \( h = 14 \) mm was selected as a constant parameter, common to all panel designs, as one of the main objectives was to reduce the structural weight, and because higher stringers would decrease the space available for other components of the fuselage, as well as the space for the luggage and passengers.
Table 2: Geometry and dimensions of the panel designs I, C, J and Ω

<table>
<thead>
<tr>
<th>Panel design</th>
<th>Geometry / Dimensions [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>32</td>
</tr>
<tr>
<td>C</td>
<td>14</td>
</tr>
<tr>
<td>J</td>
<td>14</td>
</tr>
<tr>
<td>Ω</td>
<td>12.5</td>
</tr>
</tbody>
</table>

3.2. FE mesh

The skin and stringers were discretized with 4 node conventional (2D) shell elements with reduced integration, denoted by S4R in Abaqus designation. When a conventional shell element is used, the geometry and the degrees of freedom (DOF) are associated with the reference surface, which is, by default, coincident with the shell's midsurface (in Abaqus/CAE [4]). The reference midsurface is defined by the shell’s nodes and normal definitions. In this work, the midsurface shell approach was used for all nonlinear simulations.
Each node of S4R possesses six DOF, three displacements and three rotations. The S4R element uses linear interpolation for the coordinates, displacements and rotations and only 1 integration point to compute the constitutive response of the individual element. The in-plane strains are obtained from the displacement field, whereas the stresses and transverse shear strains are calculated at the middle of the edges and interpolated to the integration point. The S4R is one of the most versatile shell elements and allows achieving the highest accuracy and a relatively low computing time (e.g. in comparison to the S4, which uses 4 integration points and makes the element computationally more expensive).

In general, the adhesive layer between the skin with the stringers was modelled with 8 node (3D) solid elements to capture the potential delamination at the interface between the skin and the stringers, caused by the damage of the adhesive. The FE applied in the adhesive depended on the damage model (DM) used.

An 8 node 3D cohesive element, COH3D8 in Abaqus nomenclature, was applied in the DM-HC (the damage models implemented in this work are described in section 3.3) at the interface between the skin and the stringer flange. The initial geometry of the cohesive element is defined:

- By the magnitude of the initial constitutive thickness, which was determined using the nodal coordinates;
- By the stack direction, in which the top and bottom faces of each cohesive element were specified.

In the models with DM-HX (see section 3.3) the 8 node linear hexahedral (“brick” - 3D) solid element, C3D8 in Abaqus designation, with full integration, was used to model the adhesive layer. These elements have 8 integration points as shown in Figure 23.

![Figure 23: C3D8 elements with full integration](image)

The first version of the mesh of panel design T5 comprised an approximate global size of 2 mm for the FE, in all geometric parts, and therefore, a total of 79820 elements. This version was rapidly rejected, since each increment, in the nonlinear analysis, required more than 5 minutes to be completed in the models in which the DM-H was applied. All models ran on a 2.5 GHz intel core i7 6500 U using the Newton’s Raphson method (Abaqus/Standard). A second mesh density of the same panel was attempted using a global size of 5 mm for the FE, thus comprising a total of 30420 elements. This latter version was, likewise, very time consuming, particularly when the damage models were incorporated in the analysis.
In order to achieve similar results to those obtained in simulation of COCOMAT panel D1, which had 7956 elements [8], a third model (mesh) was created. The FE mesh contained different densities across the panel’s width and varied with the applied DM. All DM comprised 3 elements along the stringer blade and 10 elements between the stiffeners. The DM models that did not include damage of the adhesive were modelled with 6 elements along the stiffener flange and adhesive, whereas the DM-HC and DM-HX models comprised 10 elements along the same area, since several convergence issues appeared when a fine mesh was not used in the adhesive. In the longitudinal direction, 78 elements were used in order to obtain almost square-shaped elements. An illustration of the mesh used (with 6 elements in the stringer flange) is shown in Appendix B. The FE model parameters of all designs of the T5 panel are presented in Table 3.

<table>
<thead>
<tr>
<th>Table 3: FE model parameters of all designs (Panel T5)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>COCOMAT D1</strong></td>
</tr>
<tr>
<td><strong>Number of elements</strong></td>
</tr>
<tr>
<td><strong>Number of nodes</strong></td>
</tr>
<tr>
<td><strong>Number of elements between flanges</strong></td>
</tr>
<tr>
<td><strong>Number of elements along the blade height</strong></td>
</tr>
<tr>
<td><strong>Number of elements along the stringer flange</strong></td>
</tr>
<tr>
<td><strong>Number of elements in the longitudinal direction</strong></td>
</tr>
</tbody>
</table>

Both FE meshes generated in this work were more refined than the one used in the D1 version of the panel modelled by the COCOMAT researchers. Furthermore, according to Winzen [8], the shortest computational time that was used to complete the analysis was 15 hours and 53 minutes, with the only failure mechanism being implemented in the adhesive layer connecting the skin and the stringers. However, it must be taken into consideration that the analysis of the D1 model was performed in 2008, and nowadays more powerful and recent CPU processors allow the utilization of more refined meshes.

Additionally, a convergence study was performed to examine if the two FE meshes were adequate. Hence, the model with the coarser mesh (total of 11310 elements) but without any form of damage (the damage evolution variables associated to Hashin’s failure criteria were removed from the analysis), was chosen, as well as the second version of the same panel with a global size of the
elements of 5 mm (total of 30420 elements). Figure 24 shows the load vs. axial shortening curves of the two models with mesh variations.

![Figure 24: Convergence study with two mesh variations](image)

As it can be seen, there is a high similarity between both curves and the differences can considered negligible. Therefore, the meshes developed in this work regarding the panel design T5 were considered adequate for further investigations.

Similarly, the mesh density of the additional panel designs (T4, T6, I, J and Ω), as well as an illustration of the respective meshes, are given in Appendix-B. For these models, an individual mesh convergence study was not made, but similar mesh seeding and element types were applied, assuming an equivalent sensitivity to the number of FE as that of the first panel analysed.

### 3.3. Material properties and damage models

In this section the material properties for the CFRP prepeg IM7/8552 UD, the carbon/epoxy composite material used in the skin and stringers are described. Additionally, the material data of the adhesive layer is also given. Although the nominal values of the properties of CFRP and adhesive are provided by the producer (Hexcel), several tests were performed by the COCOMAT partners of the DLR to characterize and measure that data. In this work, the same material properties as the ones used in the COCOMAT project were used throughout.

As explained in section 2.2, shell elements are defined under plane stress conditions. Thus, only the values of $E_1$, $E_2$, $\nu_{12}$, $G_{12}$, $G_{13}$ and $G_{23}$ are required to define an orthotropic material. $G_{13}$ and $G_{23}$ are included because they may be required for modelling transverse shear deformation in a shell [4]. The material data of each CFRP IM7/8552 lamina/ply are given in Table 4. The material behavior was assumed to be linear elastic up to the initiation of damage, which was determined considering the strength properties, also included (Table 4), and the Hashin criteria, described in section 2.3.2.1.
Table 4: Material properties for CFRP prepeg IM7/8552 UD [8]

<table>
<thead>
<tr>
<th>Elastic Properties</th>
<th>Strength parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_1$ [MPa]</td>
<td>147 000</td>
</tr>
<tr>
<td>$X_T$ [MPa]: tensile strength in fiber direction</td>
<td>2715</td>
</tr>
<tr>
<td>$E_2$ [MPa]</td>
<td>11 800</td>
</tr>
<tr>
<td>$X_C$ [MPa]: compressive strength in fiber direction</td>
<td>1400</td>
</tr>
<tr>
<td>$v_{12}$</td>
<td>0.28</td>
</tr>
<tr>
<td>$Y_T$ [MPa]: tensile strength in transverse direction</td>
<td>56</td>
</tr>
<tr>
<td>$G_{12}$ [MPa]</td>
<td>6 000</td>
</tr>
<tr>
<td>$Y_C$ [MPa]: compressive strength in transverse direction</td>
<td>25</td>
</tr>
<tr>
<td>$G_{13}$ [MPa]</td>
<td>6 000</td>
</tr>
<tr>
<td>$S_L$ [MPa]: shear strength</td>
<td>101</td>
</tr>
<tr>
<td>$G_{23}$ [MPa]</td>
<td>4 000</td>
</tr>
<tr>
<td>$S_T$: inter-laminar shear strength</td>
<td>131</td>
</tr>
</tbody>
</table>

The adhesive Redux 312 was also manufactured by Hexcel and its elastic (isotropic) properties and maximum strengths are given in Table 5.

Table 5: Material data of the adhesive Redux 312 [2]

<table>
<thead>
<tr>
<th>$E$ [MPa]</th>
<th>3000</th>
</tr>
</thead>
<tbody>
<tr>
<td>$G_1 = G_2$ [MPa]</td>
<td>1071</td>
</tr>
<tr>
<td>$v_{12}$</td>
<td>0.4</td>
</tr>
<tr>
<td>Max. compressive stress [MPa]</td>
<td>48</td>
</tr>
<tr>
<td>Max. shear stress [MPa]</td>
<td>38</td>
</tr>
<tr>
<td>Max. normal stress [MPa]</td>
<td>8.3</td>
</tr>
</tbody>
</table>

The laminate set-ups of the skin and stringers given in Table 1 were inserted in the composite layup editor, using a value of $t = 0.125$ mm for ply thickness and a local reference coordinate system with the primary axis (fiber orientation $0^\circ$) aligned with the panel length direction. Table 6 illustrates the local coordinate system and laminate layup for the skin, stringer flange and blade, respectively, all referred to panel design T5. The other panels were designed using the same principles.
Table 6: orientations and laminate layups of the skin and stringers

<table>
<thead>
<tr>
<th>Part</th>
<th>Local Reference Coordinate System</th>
<th>Laminate layup</th>
</tr>
</thead>
<tbody>
<tr>
<td>Skin</td>
<td></td>
<td>![Skin Diagram]</td>
</tr>
<tr>
<td>Stringer Flange</td>
<td></td>
<td>![Stringer Flange Diagram]</td>
</tr>
<tr>
<td>Stringer Blade</td>
<td></td>
<td>![Stringer Blade Diagram]</td>
</tr>
</tbody>
</table>

Regarding the different modelling approaches pointed out in the previous section, three different DM were implemented (Table 7). DM-H includes Hashin’s damage initiation criteria and a damage evolution law for the composite structure to model intra-laminar failure of the CFRP parts (skin and stiffeners), and no damage in the adhesive layer. The other two models incorporate the same damage model for the composite but also damage initiation and evolution of the adhesive by means of two distinct approaches: (i) DM-HC comprises the cohesive element technology applied at the interface of the skin with the stringers to simulate adhesive failure and (ii) DM-HX incorporates the XFEM with VCCT for modelling adhesive failure based on fracture mechanics.
The criteria developed by Hashin [19] were applied to evaluate the initiation of fiber rupture and kinking, matrix crushing and cracking and fiber-matrix shear failure. To allow for damage evolution, as explained in section 2.3.2.1, the critical energy release rate, $G_c$, also known as fracture energy, was specified for each failure mode. The values of $G_c$ of the CFRP IM7/8552 material were taken from the literature and are presented in Table 8.

### Table 8: Fracture energies of the CFRP IM7/8552 [5] and of the adhesive Redux 312 [27]

<table>
<thead>
<tr>
<th>CFRP IM7/8552</th>
<th>Adhesive Redux 312</th>
</tr>
</thead>
<tbody>
<tr>
<td>$G_{IC}$ [J/m²]</td>
<td>220</td>
</tr>
<tr>
<td>$G_{IIc}$ [J/m²]</td>
<td>630</td>
</tr>
</tbody>
</table>

Only the critical energy release rate associated to mode I, $G_{IC}$, is relevant to intra-laminar failure. It was assumed that all failure modes have the same fracture energy. Consistent units must be used from the beginning of the modelling process, so the value of $G_{IC}$ must be converted to N/mm. Therefore, the fracture energy $G_{IC} = 0.22 \, N/mm$ and the strength values of the CFRP material given in Table 4 were taken as inputs for the inclusion of ply damage mechanisms using Hashin’s criteria. Additionally, in order to overcome convergence difficulties, the viscous regularization scheme was activated using a viscosity coefficient equal to $10^{-3}$ in all damage modes.

The implementation of cohesive elements to model adhesive damage and crack propagation was described in section 2.3.2.3. The response of the cohesive elements in the model was specified as a traction-separation law through the cohesive section definition. The elastic properties of the cohesive layer (given in Table 5) were defined using uncoupled traction-separation behavior. Both shear moduli ($G_1$ and $G_2$) were calculated by means of,

$$G_1 = G_2 = \frac{E}{2(1+\nu)}$$  \hspace{1cm} (3-1)

where $E$ and $\nu$ are the Young’s modulus and Poisson’s ratio of the adhesive material, respectively.

The maximum nominal stress criterion (MAXS) was selected for damage initiation in the cohesive elements, with strength values (maximum values for the normal and shear stresses in the cohesive layer) given in Table 6. As pointed out before, the damage evolution of cohesive elements defined in terms of traction-separation is described by the fracture energy (or the critical energy release rate).
Typical values for the fracture energy of adhesives are available in the literature or determined from Double Cantilever Beam (DCB) tests, used for mode I, and End Notched Flexure (ENF) tests, used for mode II. Values of fracture energies for normal (mode I) and shear (mode II and mode III) modes were taken from Lemanski et al. [27] and are presented in Table 8. The adhesive material used in [27] was not exactly the same as the one used in this work (Redux 810 instead of Redux 312), but experimental results presented in [27] suggested it was not a critical parameter. The mixed-mode damage evolution law was based on the Benzeggagh-Kenane criterion, with power $\eta = 4.6$, which is commonly used for modelling adhesive failure in composite fuselage structures [17].

The XFEM analysis is only available for 3D geometric parts. Since no initial crack is present in the structure, the XFEM cannot be used alone, i.e., without the specification of a crack initiation criterion. The strain energy release rates at the crack tip were calculated based on the VCCT. The next steps were followed:

1) The fracture criterion based on the VCCT method was selected as an interaction property in association with an XFEM crack;
2) The crack plane normal direction was specified: the maximum tangential stress (MTS) direction was used as the default normal direction for the crack plane;
3) The BK criterion with power $\eta = 4.6$ was used as the mode-mix formulae with critical energy release rates of the adhesive interface (Table 8);
4) As no initial crack is present, damage initiation was specified in the material property definition using the MAXS criterion with the strength values used in the cohesive zone model approach. Hence, VCCT becomes active when damage initiation criteria are met, and a crack appears and propagates according to XFEM.

The simulation using XFEM with VCCT caused some complications as only one crack initiated and propagated in the selected 3D region, and thus modeling multiple cracks could only be achieved by creating partitions in the whole model. A previous analysis was firstly conducted in order to anticipate and to discover the potential regions of crack growth. Afterwards, a total of 50 partitions were created in the 5 adhesive layers, which are depicted in Figure B-1 (Appendix-B), to improve the accuracy of this damage model (DM-HX).

3.4. Boundary and loading conditions

Since all models comprised conventional shell elements with the middle surface as their reference surface, a modelling strategy had to be followed: the shell elements were arranged with the original distance of their thickness to the solid elements, which means that the skin and the stringer flange surface were situated apart from the adhesive’s bottom and top surface, respectively, by a distance corresponding to half of their thickness, 0.50 mm and 0.75 mm, respectively. In this way, the skin/stringer flange and the adhesive could not penetrate each other. The FE model of the skin-stringer connection is depicted in Figure 25. The skin and flange were connected to the adhesive by implementing a surface-based *TIE constraint in Abaqus. The master surface was chosen as the surface with the coarser mesh, for better accuracy (skin and flange), whereas the bottom and top
surfaces of the adhesive were selected as slave surfaces. For these models, meshes with matching nodes were employed to the regions around the stringers. By using the *TIE constraint each node on the slave surface was constrained to have the same motion (translations and rotations) as the node on the master surface to which it was closest. Abaqus accounts for shell thickness as it allows a gap to exist between the underlying tied surfaces. It will automatically reposition the slave nodes to be tied in the initial configuration to resolve those gaps and without causing any additional strain, as illustrated in Figure 26 (e.g. for the model with damage of the adhesive). Lastly, the connection between the stringer flange and blade was implemented using the same approach but with a node-to-surface formulation.

The postbuckling behavior of the panels, either in compression or bending, is highly sensitive to the applied boundary conditions (BC). Therefore, the experimental BC, defined in the experimental tests performed by COCOMAT partners described in Chapter 2, must be represented in the FE model. Across all panels, the BC and the applied load/displacement depended on the nature of the analysis (linear buckling, nonlinear compression or bending). An overview of the applied BC is given in Figure 27.

The fixed (clamped) side of the panel is marked by the blue edge (region A), which has all 6 DOF restrained. The right and left sides of the panel (region B) were set free, according to the experimental tests. These BC were common to all load cases.
At both ends, the first 60 mm in length (region C) were encased in resin to restrict out-of-plane movement and all rotations, in the experimental testing, as well as to ensure an even application of the applied load. This means that all panels have a free length of 660 mm instead of 780 mm (as presented in Table 1). For the linear buckling and postbuckling analysis of the panels in compression, all DOF were set to zero in this region, exception made to the axial displacement (z-direction in the global coordinate system). On the other hand, in the analysis of the panels subjected to bending, the in-plane translations in both the axial (z-direction) and transverse (x-direction), in region C were allowed, to ensure a more conservative and realistic solution.

The end loadings were applied, depending on the analysis or load case, using a prescribed concentrated force, displacement or rotation (region D). To ensure that all nodes located at the panel ends (marked in green) had the same displacement/rotation, the loaded edge had to possess a rigid body motion. That was achieved by applying either a rigid body or coupling constraint. A reference point (RP) located at the centre of curvature was assigned to the rigid edge, as illustrated in Figure B-3 (Appendix-B). The relative positions of the nodes remained constant throughout the simulation and their motion were coupled to the motion of the reference point, which means they had the same translational and rotational DOF.

The load was applied to the RP, as follows:

- **Linear buckling/eigenvalue analysis**: an axial concentrated force \( P = -1 \, N \) was applied in the RP and, through the rigid body definition, to all the nodes located at the loaded edge;

- **Compression analysis**: a prescribed displacement of \( u_z = 4 \, mm \) (DOF 3) was applied to the RP, which is rigidly connected to the entire row of nodes at the end of the panel;

- **Bending analysis**: a prescribed rotation \( |\theta_y| = 0.015 \, rad \) (DOF 5) with respect to the y-axis was applied to the RP. The motion of the end nodes is coupled to the motion of the RP.

### 3.5. Methods of analysis

In all panels, a linear buckling/eigenvalue analysis was firstly conducted to extract eigenvalues (buckling loads) and buckling modes. The latter were subsequently used in the nonlinear analysis as imperfections. On the other hand, a linear buckling analysis provides the critical buckling load, which will be also addressed in chapter 4 (numerical simulations and evaluation).

To perform an eigenvalue analysis, a linear buckling perturbation step was created. The eigensolver Lanczos was chosen (it is faster than the alternative subspace solver) and 200 eigenvalues were requested.

The geometric imperfections were introduced to all nonlinear models and were based on the first 3 buckling modes extracted from the linear buckling analysis, as the lowest buckling modes are considered to provide the most critical imperfections. Thus, the amplitudes of the buckling modes were scaled so that the imperfections corresponded to 10%, 5% and 2.5% of the shell thickness, and then
added to the original coordinates. This process was implemented by rewriting the keyword editor in the nonlinear model with the following command lines:

```
*IMPERFECTION, FILE=linear_buckling, STEP=1
  1,0.1
  2,0.05
  3,0.025
```

The nonlinear analysis (compression and bending) for all panel variations was carried out with the implicit solver provided in Abaqus/Standard based on Newton's Raphson method. To account for geometric nonlinearity, the *Nlgeom option had to be activated. A full Newton-Raphson procedure was applied, with incrementation parameters presented in Table 9, being adopted.

<table>
<thead>
<tr>
<th>Nonlinear analysis</th>
<th>Initial increment size</th>
<th>Minimum increment size</th>
<th>Maximum cut-backs allowed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compression</td>
<td>0.001</td>
<td>$10^{-17}$</td>
<td>30</td>
</tr>
<tr>
<td>Bending</td>
<td>0.0001</td>
<td>$10^{-17}$</td>
<td>30</td>
</tr>
</tbody>
</table>

The suggested initial increment size was 0.1% of the prescribed displacement for the postbuckling analysis of the panels in compression. For the analysis of the panels subjected to bending, an initial increment size was 0.01% of the total rotation was chosen in order to yield, in the first increment, an equivalent axial displacement, as that of the compressive load case, on the outer nodes of the loaded edge (where $u_z$ is maximum). As pointed out in section 2.1, Abaqus/Standard uses automatic incrementation control to choose the size of the subsequent increments.

In order to assist with convergence issues, numerical damping was incorporated into the analysis. The automatic stabilization scheme was activated using a damping factor of $2 \times 10^{-6}$. This value was chosen over the default parameter of $2 \times 10^{-4}$ to achieve similar results as the ones obtained by Degenhardt et al. [2]. Further parametric studies were performed to evaluate the influence of this parameter in the nonlinear analysis.
4. Results and Discussion

This section presents the numerical results of the analyses of the FE models developed throughout this work, presented in Chapter 3. Firstly, the linear buckling analyses results of all panel designs are shown. Secondly, the numerical results of the models of panel design T5 subjected to compression are assessed. These include:

- the numerical results of model without damage of the materials and its validation by comparison with the corresponding results obtained by the COCOMAT project researchers;
- the numerical results of the models with the implementation of 3 different damage models and their comparison with the experimental results: DM-H, DM-HC and DM-HX as described previously in section 3.3.2.

Thirdly, the results of the postbuckling analysis extended to the additional panel designs, which includes the comparison of the load-shortening curves and the corresponding conclusions, are shown. Finally, the results of the analysis of all the panel configurations subjected to bending is presented.

4.1. Buckling of panels under compression

This type of analysis was performed in order to obtain the critical buckling loads and mode shapes. In this work, the latter are the most valuable outcomes of the linear analysis, because in general they predict the buckling modes that are most likely to occur in the structure. These were also used in the postbuckling analyses of the panels as initial imperfections, in order to study the sensitivity of the panels to this parameter.

The buckling loads are determined through the linear analysis by multiplying the eigenvalues with the nominal axial load applied in the loaded edge. Since a value of $P = -1 \, N$ was applied to the reference point (RP) rigidly connected to the load edge, the buckling loads are directly given by the eigenvalues.

Regarding the results of panel design T5, the first local buckling load ($P_{\text{cr,local}}$) (1st mode) was predicted to be $P_{\text{cr,local}} = 44.7 \, kN$. The values of $P_{\text{cr,local}}$ of the remaining panel designs are shown in Table 10.
Table 10: Results of the linear analysis all panel designs

<table>
<thead>
<tr>
<th>Panel Design</th>
<th>1st local buckling</th>
<th>$P_{cr,local}$ (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T5$</td>
<td>1</td>
<td>44.70</td>
</tr>
<tr>
<td>$T4$</td>
<td>1</td>
<td>30.56</td>
</tr>
<tr>
<td>$T6$</td>
<td>1</td>
<td>73.61</td>
</tr>
<tr>
<td>$I$</td>
<td>1</td>
<td>56.97</td>
</tr>
<tr>
<td>$C$</td>
<td>1</td>
<td>32.90</td>
</tr>
<tr>
<td>$J$</td>
<td>1</td>
<td>50.85</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>1</td>
<td>51.03</td>
</tr>
</tbody>
</table>

As pointed out in section 3, the geometrical imperfections were introduced in all models later studied by means of fully nonlinear analyses and were based on the first 3 buckling modes extracted from the linear buckling analysis, which comprised several longitudinal buckling waves between the stiffeners. Section 4.2.2 will provide a parametric study to examine the influence of imperfections on those results. The first buckling mode of all panels is illustrated in Table C-1 (Appendix-C).

4.2. Postbuckling of reference panel T5 under compression

4.2.1. Model without damage

Figure 28 exhibits the load-shortening curve of panel design T5 without damage of the materials included. The load ($P$) was obtained by requesting and summing the values of the reaction force in the axial direction (RF-3) for the whole collection of nodes that have all DOF restrained (located at the fixed/clamped side of the panel). The axial shortening ($u_z$) was given directly by the displacement of the reference point (RP).

The results without damage exposed the typical behavior of compressed stringer-dominant panel designs, where the 3 remarkable load levels previously described in chapter 2 can be easily distinguished. The structure under axial compression developed specific mode shapes and deformed into minimum potential energy configurations. When the structure reached the first local buckling (load level A), the load was redistributed in the whole domain and there was a slight “knee” (small decrease of the stiffness) in the load-shortening curve. After that, the load was increased until the global buckling was reached (load level B), in which the stringers buckled and a high decrease of the axial stiffness occurred. From this point on, since the skin could only carry a small fraction of the applied
load, the stringers started to carry the additional load. In the end, the collapse of the panel (load level C) arose with a sharp reduction of the axial stiffness and a drop of load for increasing displacement.

![Diagram of Load vs. Axial Shortening](image)

**Figure 28: Load-shortening curve for panel design T5 without damage**

Table 11 presents the load and displacements for the 3 referred load levels, as well as an illustration of deformed shape of the panel for those stages. It must be noted that, since there is no damage involved, the collapse of the structure occurred by sudden failure due to a strong mode switch, more noticeable in the vicinity of the middle stiffener, at a load level of about 126 kN. Buckling itself can result in failure when the postbuckling behavior of the structure is unstable. In this case, for increasing shortening, the only way equilibrium can be established is if the load decreases. Therefore, the maximum attained load ($P_u$), or the maximum compressive force transmitted through the panel, was $P_u = 126$ kN for the model without damage.

The incorporation of damage in the FE models and the evaluation of the numerical results yielded by their analysis required a primary numerical validation of the results of the models without damage. This was made by means of comparing the latter with the ones obtained in the COCOMAT project. For that purpose, Figure 29 shows the load-shortening curve of the models developed in both works (COCOMAT and this work). Up to the first local buckling there was excellent agreement between the two curves, which likewise presented similar slopes up to an axial shortening of about 1 mm. The first global buckling load predicted by the analysis of the model developed in this work was slightly higher than the one attained by its COCOMAT counterpart and for a slightly higher axial shortening (1.2 mm rather than 1.0 mm). Nevertheless, the predicted postbuckling stiffness was practically identical, whereas the displacement at collapse was smaller ($u_z = 3.15$ mm instead of $u_z = 3.45$ mm), which is considered a better result because it was closer to the, measured, experimental displacement at collapse. The predicted $P_u$, which is a very important parameter in these panel designs, was almost the same ($P_u = 126$ kN in this work instead of $P_u = 125$ kN in the COCOMAT project), which is also a satisfactory outcome. In spite of the geometrical data, material properties and laminate set-up of skin and stringers adopted in both models being very similar, the slight differences
in the results predicted by the two numerical analyses could be associated with some different modelling approaches. These are the connection methods between nodes/surfaces (e.g. *Tie constraints over MPC links), mesh density and artificial damping parameters applied in the numerical solvers. Nevertheless, overall, the numerical model developed was considered validated.

Table 11: Typical behaviour of stringer stiffened panels and presentation of 3 load levels

<table>
<thead>
<tr>
<th>Load Level</th>
<th>Illustration (out-of-plane displacements ($u_z$))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load Level</td>
<td>$u_z$ (mm)</td>
</tr>
<tr>
<td>A - Local</td>
<td>≈ 0.45</td>
</tr>
<tr>
<td>B - Global</td>
<td>≈ 1.20</td>
</tr>
<tr>
<td>C - Collapse</td>
<td>≈ 3.15</td>
</tr>
</tbody>
</table>

Figure 29: Load-shortening curve of the panel design T5 without any type of damage
When comparing the numerical load-shortening curve of the compressed stiffened panel with the experimental curve presented in section 2.4, it can be concluded that the solution gave a good prediction of the panel stiffness up to the global buckling load. Beyond that point, because damage was not considered in the model for calibration purposes, the differences naturally became larger. This emphasizes the necessity of implementing some form of material damage in the FE models.

4.2.2. Influence of imperfections and damping factor

This sub-section focuses on two parametric studies performed in this work in order to evaluate the sensitivity of the panel to (i) geometrical imperfections of the skin and to (ii) the value of the damping factors applied in the nonlinear solver. These studies were performed on the model without damage, as this one required less time to complete the analysis in comparison with the models in which damage mechanisms were employed.

As pointed out in previous sections, the first 3 eigen modes were extracted from the linear buckling analysis and used in the subsequent nonlinear analysis as superimposed artificial imperfections. The amplitudes of the buckling modes were scaled so that the imperfections corresponded to 10%, 5% and 2.5% of the shell thickness. To investigate the influence of imperfections, two additional analysis were performed, one with the nominal panel, i.e., without imperfections and the other with an imperfection amplitude for the first mode of 50% of the thickness (25% and 12.5% for the second and third modes). The results are shown in Figure 30, in which the legend of the curves refers to the first buckling mode.

The introduction of an imperfection with a value of 10% of the thickness had a negligible effect in the initial axial stiffness as well as in the prediction of global buckling, postbuckling region and collapse of the imperfect model in comparison with those of the nominal model. On the other hand, the
imperfection with a value of 50% of the thickness conducted to a strong reduction of the load-carrying capacity of the panel for an axial displacement of about 2.1 mm. This was due to a sudden asymmetric mode switch in the vicinity of the middle stiffener, in contrast to the constant slope of the previous models. After that, the slope of the curve was approximately constant up to 4 mm of shortening. This influence was expected, since the highest imperfection, of 50%, associated to first buckling mode consists of a significant geometrical deviation from the perfect structural shape, most likely to be unrealistic, so it was not considered in further simulations.

Summing up, all models studied by means of non-linear analyses were ran with imperfections displaying maximum value of 10% of the thickness, which had a minimal effect on the deformation progression and load-carrying capacity, as well as on other panel characteristics.

The effect of the user-defined damping factor in the nonlinear analysis conducted by Abaqus/Standard is depicted in Figure 31. Four values were tested for this purpose: $2 \times 10^{-7}$, $2 \times 10^{-6}$, $2 \times 10^{-5}$ and $2 \times 10^{-4}$. It can be concluded that the highest damping factor $DF = 2 \times 10^{-4}$ led to a significant overestimation of the load-carrying capacity of the panel in the postbuckling region, whereas the damping factor $DF = 2 \times 10^{-5}$ did not produce accurate results regarding the typical panel behaviour observed in the experimental programme. The value $DF = 2 \times 10^{-7}$ predicted a similar stiffness of the panel up to the global buckling point as well as up to an axial shortening of $u_z = 2.2$ mm, but from this point on, compared with the results yielded by the application of the previous damping factor, the axial stiffness was very different. On the other hand, it must be noted that the computational time required for the solution to be obtained increased substantially with the lowest values of the damping factor. Hence, the damping factor $DF = 2 \times 10^{-6}$, also chosen by Degenhard et al. [2], was applied in all the analysis of all the models developed in this work.

![Figure 31: Influence of the damping factor in the nonlinear analysis using Abaqus/Standard](image-url)
4.2.3. Models with damage

In this subsection, the main results of the simulations considering the three damage models described before are presented. The global deformation patterns (mode shapes) of each model are shown in section 4.2.4 for the purpose of comparison with the experimental results.

4.2.3.1 Hashin’s damage model

As pointed out before, ply failure is one of the critical damage mechanisms of composite laminates and a progressive damage model based on Hashin’s criteria was applied to represent the accumulation of damage at the ply level. It considers 4 different damage modes, namely failure of the fiber and matrix with separate mechanisms in tension and compression, and an additional fiber-matrix shear as an individual failure mode. The damage evolution is based on the fracture energy and is characterized by progressive degradation of material stiffness, leading to material failure.

Figure 32 shows the load-shortening curve of the simulation of the damage model with Hashin’s criteria (DM-H) implemented. The graph also exhibits the load-shortening curve of model without any type of damage, for a better evaluation of the effect of DM-H in the panel behavior.

![Figure 32: Load-shortening curve for panel design T5 with DM-H](image)

As shown in appendix A, the output variables HSNFTCRT, HSNFCCRT, HSNMTCRT and HSNMCCRT indicate whether the damage initiation criteria in the 4 distinct damage modes have been satisfied or not. These are governed, respectively, by equations (2-11) to (2-14) and depend on the laminate material strengths. When any of the indexes reaches 1.0 it means that the respective damage initiation criterion has been satisfied and damage has begun for a given element of the skin, stringer flange or blade. Once any of the damage initiation criteria is satisfied, the values of stiffness are reduced, and the response of the material is computed from equation (2-18), where 5 damage variables (DAMAGEFT, DAMAGEFC, DAMAGEMT, DAMAGEMC and DAMAGESHR) intervene, for each mode, and quantify the degree of damage a given element. If one damage variable reaches a value of 1.0, it means that the element is fully damaged/degraded.
Prior to the onset of global buckling, both models predicted a similar initial axial stiffness, with the corresponding curves almost coincident, as seen in Figure 32. Even though both models present similar load-shortening curves, it doesn’t mean that damage was not present in the model in which DM-H was included. In fact, damage initiation was predicted in the skin matrix, in plies oriented at 90°, at rather small values of axial shortening, of about $u_z = 1.0$ mm. This was indicated by the output variable \textit{HSNMCCRT}, which referred to the matrix compression mode and physically means that the matrix in the skin crushed for the denoted areas (damage mechanisms are described in section 2.3.1). Summing up, the first ply failure occurred at the top and bottom layers of the skin (Figure 33), which have the fibers oriented at 90° with respect to the applied loading, for an axial shortening $u_z = 1.0$ mm, a corresponding reaction force $P = 82$ kN and maximum out-of-plane displacement (y-direction) $|u_y| = 2.55$ mm. For $u_z = 1.05$ mm, the elements that were fully damaged (quantified by the output variable \textit{DAMAGEMC}) depended not only on the orientation of the fibers, but also on the position of the layers relative to the topmost one, as the top ply of the skin layup presented already a considerable number of completely degraded elements, whilst in the bottom ply of the skin they were scarce (Figure 34).

![Figure 33: Damage initiation (matrix crushing mode) for the 90° layers of the skin matrix in (a) bottom ply and (b) top ply for $u_z = 1.0$ mm](image1)

![Figure 34: Degree of damage (matrix crushing mode) for the (a) bottom ply and (b) top ply of the skin for $u_z = 1.05$ mm](image2)

The global buckling occurred for a smaller value of load in comparison with the model with no damage included, in particular at a load $P = 90.6$ kN and an axial shortening $u_z = 1.17$ mm. The fact that a considerably number of elements had their elastic properties degraded can explain the slight reduction in the global buckling load compared to the results of the model without damage. This load level caused bending of the outer stiffeners, and thus the bending stresses and strains increased in the
outer layers. This means that the maximum bending stresses are either compressive or tensile on the inner and outer sides of the buckling waves peaks, respectively. At this stage, more damaged areas could be observed in the matrix of the skin, as shown in Figure 35, in both top and bottom plies of the skin layup, where damage has spread to the closest plies, which are oriented at 45°. The maximum out-of-plane displacements were located at the middle of the skin between the outer stiffeners and the adjacent ones (similar to what occurred in the model without damage), with a maximum magnitude $|u_y| = 6.96$ mm (Figure 36).

Summing up, there was a premature failure in the form of matrix crushing at the outer and inner layers of the skin as they have their fibres oriented perpendicular to the loading direction. When the load was increased, damage propagated to the adjacent layers as the initially damaged layers could not carry any additional load. Therefore, the subsequent layers also failed in the form of matrix crushing.

This failure mode also appeared in the stringers, with more intensity in the flange of the middle stringer, as depicted in Figure 37. However, the load-shortening curve shows that this has a rather negligible effect on the load-carrying capacity of the structure.
Moreover, the damage mode associated with fiber-matrix shear failure, and given by the damage variable \textit{ DAMAGESHR}, followed the same path as the matrix crushing (compression) mode, with almost the same predicted damaged areas, as shown in Figure C-2. Regarding this failure mechanism, the top and bottom layers of the skin were also more susceptible to damage initiation than the inner layers, as they have their fibers oriented at 90º with respect to the principal axis.

The other failure mode of the matrix (cracking) appeared in the skin, in the stringer flange and in the stringer blade for axial shortenings of $u_z = 1.52$ mm, $u_z = 2.13$ mm and $u_z = 2.25$ mm, respectively. The number of FEs that had their matrix entirely cracked after collapse is shown in Figure C-3, for multiple section points. Nevertheless, this damage mode did not have much influence of the load carrying capacity of the panel as well.

It can be concluded that the two matrix damage modes (crushing and cracking) do not have a major impact on the global behaviour of the stiffened composite structure, even for a significantly number of fully degraded elements. Hence, one early conclusion can be anticipated: the decrease of the load-carrying capacity of the structure with this implemented damage model in comparison with the case with no damage involved was induced by the appearance of fiber damage.

The fibers resist the majority of the applied loads. Consequently, under axial compression, failure in the form of fiber micro-buckling (fiber compression/kinking) was expected to occur initially at the layers with the fibers oriented at 0º with the loading axis, as these layers carried a higher percentage of the applied load, and thus they are more susceptible to fail under this mode. In fact, as depicted in Figure 32, the panel started to collapse for an axial shortening of about $u_z = 2.5$ mm. At this point, fiber kinking appeared in the blades of the outer stringers 1 and 5 (the numbering of the stringers is also given in Appendix-C), at the plies oriented at 0º, as shown in Figure 38. There, the damage variable associated to the fiber compression failure mode for one of the outmost layers oriented at 0º can be observed, where two FEs are totally degraded (blade 5) and one partially damaged (blade 1). At this point the layers of the stringer blades oriented at 45º and -45º have their fibers fully intact.
When the axial shortening was further increased \( (u_z > 2.5 \text{ mm}) \), this form of fibre failure spread to more elements and was transferred to the adjacent layers oriented at 45° and -45° with the load direction. This is shown in detail in Figure C-4 for an equilibrium configuration far beyond collapse (for an axial shortening of \( u_z = 3.3 \text{ mm} \)).

On the other hand, the fibre kinking failure mode also appeared in the middle stiffener flange. As depicted Figure 39. The torsion of the middle stiffener under increased compression caused the appearance of fibre kinking there for an axial displacement \( u_z = 3.31 \text{ mm} \). Nevertheless, this has not contributed to the onset of collapse (it began at \( u_z = 2.5 \text{ mm} \) as already pointed out), but may have caused a larger loss of axial stiffness.

Finally, the last remaining failure mode, fiber fracture (or fiber tension/rupture) only appeared in the skin matrix, first in the topmost ply oriented at 90° with the loading direction (for \( u_z = 3.27 \text{ mm} \)) and immediately after in the bottom ply of the skin layup, also oriented at 90° with the loading direction (for \( u_z = 3.29 \text{ mm} \)). Figure 40 depicts this ultimate form of failure, where fiber fracture teared the skin and
caused a "catastrophic" loss of axial stiffness and load carrying capacity. This forced the solver to end the analysis (since increments of very little magnitude and lot of cut-backs were necessary to pursue the nonlinear equilibrium path).

4.2.3.2 Hashin’s damage model with cohesive elements

This damage model combined the intra-laminar damage governed by Hashin’s criteria and cohesive elements applied to the adhesive layer. Because the influence of composite damage can be significant, as shown previously, intra-laminar damage was also included again to perform the most possible thorough and accurate analysis.

This modelling approach was, most of the times, problematic and challenging, since a fine mesh was required for the analysis to remain accurate, as well as for the several convergence issues that appeared in the deep postbuckling region to be tackled. The performed analysis did not reach the desirable magnitude of axial shortening mainly due to contact problems. Once fully degraded at all of its material points, cohesive elements were removed from the analysis and offered no resistance to subsequent penetration of the remaining components. This brought many numerical instabilities, even with the viscous regularization scheme applied to the constitutive equations. Therefore, numerous attempts to obtain convergence were made by changing the damping factors of the traction-separation laws, of the Hashin damage variables (in all failure modes) and of the numerical solver itself. On the other hand, different mesh densities were analyzed and applied in order to improve convergence, but always taking into consideration the computational time required.

Figure 41 shows the numerical load-shortening curve of the model in which DM-HC was applied, as well as the one resulting of the analysis of the model without the inclusion of any kind of damage. As it can be observed, the introduction of damage in the adhesive caused the global buckling of the panel to occur at lower values of load and axial shortening ($P = 88.2$ kN and $u_z = 1.17$ mm). After global buckling, the stiffness of this new model (with DM-HC) was slightly lower than that of the original model, up to an axial shortening of $u_z = 2.26$ mm, but this deviation was negligible. However, for an axial shortening $u_z = 2.26$ mm, failure in the form of skin-stringer debonding arose in both edges of the
middle stiffener and near the potting region in one of the inner stiffeners (stiffener 4) on the loading side. This type of failure was caused by the partial or total degradation of some elements in the adhesive, which started to fail when the nominal stress damage initiation criterion (MAXS) reached the value of 1.0 in a material point during the analysis. The damage initiation and evolution of the adhesive elements was monitored by the output variables MAXSCRT and SDEG, respectively. The quadratic nominal stress damage initiation criterion (QUADS) was also used and tested but gave similar results as those of the MAXS criterion, with the same predicted debonded areas and with an almost identical load-shortening curve.

\[ \text{Figure 41: Load-shortening curve of the model with cohesive elements in the adhesive layer (DM-HC)} \]

The predicted initiation of skin-stringer debonding, induced by the damage of adhesive elements for an axial shortening \( u_z = 2.26 \, \text{mm} \), is illustrated in Figure 42. Under further loading, the growth and number of failed elements increased promptly, with a considerable number of elements predicted to have failed at an axial shortening \( u_z = 2.36 \, \text{mm} \), as depicted in Figure 43. Once totally failed (with zero stiffness), these elements acted only as a contact region to deny any physically cross-over of the skin with the stringers. From visual inspection, it can be seen that some completely failed elements were not removed from the analysis (the ones wholly colored in red), because Abaqus only deletes cohesive elements if none of its material points are in compression. Nevertheless, the debonded areas can be straightforwardly distinguished.
The predicted separation between the skin and stiffeners caused the appearance of one other “explosive” damage mechanism, explicitly fiber kinking across the blade in one of the inner stiffeners (stringer blade 2, close to the potting region on the non-loading side, as illustrated in Figure C-5). This likely led to the collapse of the panel for an axial shortening $u_z = 2.36$ mm. At this point a strong reduction of the load-carrying capacity occurred, and it was not possible to go further towards higher values of axial shortening, mainly due to contact problems that caused severe convergence issues. The panel under this DM also displayed other composite failure mechanisms such as matrix crushing and cracking in the skin and stringer flange and blade.
4.2.3.3 Hashin’s damage model with XFEM

This damage model combined (i) the XFEM, which is entirely based on fracture mechanics’ theory and allowed to track the motion of cracks and (ii) again, due to the significant influence of composite damage, Hashin’s criteria.

Similarly to the previous DM, this modelling approach also led to various convergence problems and required a refined mesh to be adopted in order to produce accurate results. The typical requirement for a fine mesh around the crack front could not be modelled because no initial cracks were present in the structure. Therefore, the 5 adhesive layers had the same mesh density. On the other hand, the use of the Virtual Crack Closure Technique (VCCT) within XFEM (to calculate the strain energies releases at the crack tip) was studied in detail, because several parameters linked to contact properties were by default unfilled, such as (i) the direction of crack growth relative to local 1-direction, (ii) mix mode behaviour and (iii) its exponents. The maximum tangential stress (MTS) direction was used as the normal direction for the crack plane over the alternatives (normal and parallel directions) because it was recognized (after all the attempts) as the one that produced more realistic results concerning the crack propagation. Additionally, it was the one that led to best agreement between numerical and experimental load-shortening curves.

Moreover, in order to visualize the location of the crack fronts, the output variable PHILSM had to be requested, as Abaqus automatically creates an isosurface view cut based on this output. Otherwise, the cracks would not be visible. Additionally, the output STATUSXFEM was equally called, which indicates if the element is partial or completely cracked (with no traction across the crack faces).

Figure 44 shows the numerical load-shortening curve resulting from the nonlinear analysis of the model in which DM-HX was applied, as well as the one of the model without the inclusion of any kind of damage. The introduction of this DM essentially altered the structural behavior of the panel after the onset of global buckling, with a lower value of the load but a similar postbuckling stiffness up to an axial shortening of about $u_z = 1.95$ mm, being displayed. At this magnitude of shortening, the significant reduction in the load was caused by the development of a second global buckling shape in the vicinity of two of the inner stiffeners (stringers 3 and 4), which buckled inwards as shown in Figure C-6 (Appendix-C). The behavior of the panel with this DM was considerably affected by failure of the adhesive layer, as expected. This type of failure was represented by the onset and growth of cracks. Damage initiation was specified in the material property definition using the MAXS criterion and, when met, a crack appeared. Afterwards, VCCT became active and the crack growth was controlled by the rate of strain energy released at the crack tips.

Crack initiation and growth were predicted to occur at multiple locations throughout the panel, beginning when the applied shortening was $u_z = 1.63$ mm, and up to the final point at which the collapse of the panel occurred. For a shortening of about $u_z = 2.49$ mm the panel also denoted signs of fiber kinking in the middle stiffener blade, close to the potting region, which caused a slight “knee” in the load-shortening curve. Matrix crushing and cracking appeared likewise in the skin and stringers during the compression of the panel, as depicted in Figure C-7.
Figure 44: Load-shortening curve of the model with XFEM (DM-HX)

The collapse of the structure occurred for \( u_z = 2.65 \) mm, and was induced by an abrupt failure in the middle of the two outer stiffeners, 1 and 5, which developed fiber kinking in these areas as depicted in Figure 45. After collapse, the adhesive layer exhibited numerous single cracks, affecting only 1 FE, and two very long cracks underneath stiffeners 2 and 3, which resulted from crack propagation through seven and five FE, respectively. The crack fronts after collapse are shown in Figure 46 and Figure 47.

Figure 45: Fiber kinking in the stringer blades for \( u_z = 2.55 \) mm (top) and \( u_z = 2.65 \) mm (bottom)
**Figure 46**: Multiple cracks at the adhesive layer (stringer 5 is not visible)

**Figure 47**: Details (zooms) of crack propagation in the vicinity of the adhesives underneath the stiffeners 3 and 4
4.2.4. Comparison of results

The assessment of the achieved numerical results comprised the validation by comparison with the experimental results, which included the comparison of the load-shortening curves and deformation patterns or mode shapes, given in Figure 48 and Table 12, respectively.

**Figure 48: Load-shortening curves of the numerical models developed and experiment**

From Figure 48, one can see that all numerical load-axial shortening curves show a very good agreement regarding initial axial stiffness, up to the point of global buckling. However, all models predicted a higher global buckling load than that measured on the experimental test, with a relative difference of about 20%. From that point, the first model developed, i.e., the one without the inclusion of damage, was the one that most overestimated the panel load carrying capacity in the postbuckling region, which underlined the necessity of implementing damage models. On the other hand, the model that accounted exclusively for composite damage mechanisms (DM-H) overestimated the postbuckling stiffness and the value of $P_u$ and anticipated the onset of collapse, in terms of axial shortening.

It must be noted that the inclusion of numerical damping into the analysis ($DF = 2 \times 10^{-6}$), as well as the implementation of the viscous regularization scheme into the damage elasticity matrix, which used the viscous values of damage variables (viscous coefficient equal to 0.001 for all damage modes), amplified the overestimation of the global buckling loads and $P_u$. If no viscous regularization was applied, the difference between numerical and experimental values of the global buckling load would have been lower, and the numerical and experimental load-axial shortening curves would have been closer, tough the time required for the analysis to be completed would have been much larger. Additionally, convergence was compromised, in some cases, when multiple damage models were included. Therefore, the evaluation of the behavior of the panels with each damage model was made
with more emphasis in the qualitative aspect of the curves, naturally attempting to minor the differences as well, but knowing a priori that Abaqus (with the viscous regularization scheme) consistently overestimates the load-carrying capacity of the panel [4].

The model with cohesive elements applied in the adhesive layer (DM-HC) didn’t differ significantly in the progression of the load-shortening curve, compared with the remaining, up to the point where the skin began to separate from the stringers (around \( u_z = 2.26 \) mm). At that point, there was a sharp load decrease, which did not correlate well with the experiment, because for that axial shortening the load was expected to still be increasing. However, the areas of skin-stringer debonds predicted by this model compared very well with those of the experiment, though the exact location of failure at the middle stiffener was slightly different, but the agreement was considered very satisfactory.

Lastly, the model that combined the XFEM with Hashin’s damage criteria (DM-HX) was the one that resulted in the closest prediction of the load-carrying capacity determined experimentally. The progression of this numerical load-axial shortening curve also looked very similar to its experimental counterpart, although the former always lies above the latter. This model also predicted the second global buckling shape for a slightly higher axial shortening (in the vicinity of two of the inner stiffeners as described in the previous section 4.2.3.3), and collapse occurred for \( u_z = 2.65 \) mm, which is a very good prediction because the tested panel collapsed for \( u_z = 2.71 \) mm.

For the comparison of the deformation patterns or mode shapes of all FE models with the experimental one, shown in Table 12, it must be noted that all panels were designed with the stringers pointing in the positive y-direction. On the contrary, the COCOMAT panels were designed upside down, i.e., with the stringer side pointing in the negative y-direction (see Figure 49). As a result, the deformation patterns were indicated with opposite colors with respect to the ones specified in COCOMAT and in the experiment, as follows:

- Inward displacement / movement towards the centre of curvature: marked in red (blue in COCOMAT and experiment)
- Outward displacement / movement away from the centre of curvature: marked in blue (red in COCOMAT and experiment)

Figure 49: front view (x-y plane) of the T5 panel

The panel tested experimentally underwent local buckling for an axial shortening \( u_z = 0.51 \) mm, exhibiting 15 longitudinal half sine waves between the stiffeners and a slightly asymmetric deformation pattern. The first global buckling of the panel occurred for an axial shortening of about \( u_z = 0.97 \) mm, the asymmetric deformation pattern remained visible and the outer stiffeners buckled, with one side of the panel moving away and the other side moving towards the centre of curvature. For an axial
shortening of about $u_z = 1.72$ mm, the inward global buckle grew towards the middle stiffener. Finally, the collapse of the panel occurred for a shortening of $u_z = 2.71$ mm, with a highly asymmetric deformation pattern and two global buckles being visible, the right one moving inwards (towards the centre of curvature) and the left one moving outwards. These mode shapes are shown in the first row of Table 12.

All models predicted very well the local buckling shapes of the panel (up to the global buckling point) in comparison with those measured experimentally, the exception being the DM-HX, which predicted 13, rather than 15 as in the other models, longitudinal half sine waves between the stiffeners. Oppositely, in all models the slight asymmetric pattern of deformation (observed experimentally) was not predicted.

The model without damage of the materials and the one that accounted exclusively for composite damage (DM-H) predicted similar symmetrical global buckling deformation patterns, with two inward buckles in the outer stiffeners (towards the centre of curvature). The asymmetric outward/inward deformation pattern (left/right sides of the panel, respectively) seen experimentally was again not predicted by these models.

On the other hand, the model with cohesive elements applied in the adhesive layer (DM-HC) was able to predict an asymmetric deformation pattern developing for an axial shortening of about $u_z = 1.17$ mm. At this point, an outward global buckle (marked in blue) developed between stiffeners 2 and 3 and, under further loading, it moved to the left and fixed underneath stiffener 2. Though the exact location of the outward buckle seen experimentally was underneath the left outer stiffener, it can be concluded that this model predicted a postbuckling deformation pattern close to that of the experiment.

The model that combined the XFEM with Hashin’s damage (DM-HX) gave, as stated before, the closest prediction of the panel behavior in terms of the progression of the load-shortening curves, and very good prediction of the point of collapse. In spite of this, the global deformation pattern differed from the experiment, as it was nearly symmetric, with one inward buckle located at the centre of the middle stiffener, instead of on the right side, as observed in the panel tested experimentally, and two outward buckles in the outer stiffeners.
**Table 12: deformation patterns at different values of axial shortening**

<table>
<thead>
<tr>
<th></th>
<th>uₓ = 0.51 mm</th>
<th>uₓ = 0.97 mm</th>
<th>uₓ = 1.31 mm</th>
<th>uₓ = 1.72 mm</th>
<th>uₓ = 2.71 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Experiment</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Model without damage</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>DM-H</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>DM-HC</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>DM-HX</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

| Model without damage | uₓ = 0.63 mm | uₓ = 1.2 mm | uₓ = 2.8 mm | uₓ = 3.3 mm | uₓ = 4.0 mm |
| DM-H            | uₓ = 0.68 mm | uₓ = 1.17 mm | uₓ = 2.5 mm | uₓ = 3.0 mm | uₓ = 3.3 mm |
| DM-HC           | uₓ = 0.58 mm | uₓ = 1.17 mm | uₓ = 2.12 mm | uₓ = 2.3 mm | uₓ = 2.36 mm |
| DM-HX           | uₓ = 0.57 mm | uₓ = 1.16 mm | uₓ = 1.47 mm | uₓ = 1.97 mm | uₓ = 2.71 mm |
The inability of some models to correctly capture the asymmetry observed in the experiment can be related to the following main reasons:

- Firstly, there was considerable uncertainty associated with some material parameters. For instance, there were variations on the material data of the adhesive measured by the COCOMAT partners, as numerous tests were performed to validate the data provided by the producer (Hexcel). Additionally, there is significant uncertainty related to fracture toughness values in general, as the mode II energy is difficult to determine. Moreover, there is likewise uncertainty for the parameters that govern the mix-mode behavior;

- Secondly, there was an important inter-laminar failure mechanism that was not considered in this work throughout. Delamination between the composite plies was not incorporated into the analysis because it required the definition of several layers of shell elements, one for each ply and, consequently, more FE and a huge increase in computational time-consumption. An attempt was made using 3D continuum shell elements for the first four individual plies of the skin, and cohesive elements implemented at the interface between them. The objective was to study the effect of delamination on the first four plies of the skin which had different fiber orientations. The damage of the adhesive was not considered, but DM-H was implemented. However, this analysis would require weeks to be completed with the processor used, so it was impossible to go on with this approach;

- Thirdly, the conventional shell element used (2D) did not account for the through-thickness stresses, which could have altered the prediction of the deformation mode shapes;

- Fourthly, it is possible that some errors could have influenced the experimental tests, such as an asymmetric introduction of the load and the potting region not being exactly parallel, for example. On the other hand, it is possible that the initial imperfections of the tested panel are different from the adopted ones;

- Finally, it was noted that the application of different damping parameters into the numerical solver altered significantly the deformation patterns. It was also found that the application of the damping factor $DF = 2 \times 10^{-5}$, in spite of overestimating the load-carrying capacity of the panel, predicted the exactly same deformation pattern of the experiment, as depicted in Figure 50. The reason for not pursuing all the analysis with this value of damping was that it produced a non-realistic and overestimated load-shortening curve.

![Figure 50: Deformation pattern of the T5 panel with a damping factor of $2 \times 10^{-5}$](image)
In conclusion, the different numerical results achieved for this panel demonstrated the aptitude of the adopted approaches to provide accurate predictions of the load-carrying capacity and deformation shapes, as well to predict correctly the different damage mechanisms in composite panels subjected to axial compression.

4.3. Postbuckling of different panels under compression

In this section, the postbuckling analysis of panels subjected to compression was extended to the additional six designs (T4, T6, I, J and Ω). Therefore, we study:

- the influence of the number of T-shaped stringers on the postbuckling and strength of panels (comparison between the reference panel T5 with T4 and T6) and
- the influence of the stringer shape on the postbuckling and strength of panels (comparison between the reference panel T5 with I, C, J and Ω)

Herein, all panel designs were analyzed and compared merely including the composite damage model (DM-H). This was due to the significant convergence issues and the almost tripling of the total computation time required, resulting from the implementation of damage in the adhesive layer, as well as the lack of experimental data of the other panel configurations (besides the usual T-shaped stringer panels). The same total axial shortening ($u_x = 4.0$ mm) was applied to the RP of all panels to assess and compare their structural behaviour.

Table 13 presents the main numerical results of the seven analyzed panels, specifically (i) the axial shortenings at the local and global buckling loads, (ii) the $P_u$ values (maximum axial loads that the panels can withstand) (iii), the shortening at collapse ($u_c$), (iv) the shortening between the first local and global buckling ($\Delta u_{lg}$) and (v) the shortening between the global buckling load and collapse, i.e., the post-global buckling shortening ($\Delta u_{gc}$). Collapse was considered to be the first value of axial shortening for which an abrupt decrease in the load carrying capacity of the panel was observed, caused by fiber kinking, as justified before. The breakage of the fibers typically occurred for higher values of axial shortening, resulting in a large decrease in the load, and led to catastrophic failure of the structures. However the onset of collapse was understood to occur at the beginning of the load reduction, which was typically associated to fiber kinking. The post-global buckling shortening is defined as the area between the first global buckling load and collapse. Panels with large $\Delta u_{lg}$ can resist several buckling waves arising between the stiffeners until the onset of stringer-based buckling, during which matrix crushing and cracking mechanisms can be detected in the skin. On the other hand, panels with large $\Delta u_{gc}$ are still capable of resisting further load after the stringers have buckled, and can withstand more severe damage until the onset of structural collapse. The postbuckling shortening is the sum of $\Delta u_{lg}$ and $\Delta u_{gc}$. 
The load-shortening curves of the seven panels are shown in Figure 51. Panel designs T4 and T6 have the same arc length as panel T5, but own four and six stringers, respectively, instead of five. As these panels were taken as sections with approximately 32° of equivalent cylinder design with a radius \( r = 1000 \text{ mm} \), it implies that the panel design T4, T5 and T6 would represent cylindrical fuselage shapes with 45, 56 and 67 stringers, respectively.

Panel design T4 turned out to be the one presenting the lower stiffness of the three T-shaped panel versions, as expected, with the lowest value of \( P_u = 81.9 \text{ kN} \), predicted at the point of global buckling \( (u_z = 1.56 \text{ mm}) \). This panel presented a larger \( \Delta u_{lg} \), but also a smaller \( \Delta u_{gc} \) than those of panel T5. The appearing of damage mechanisms such as matrix crushing and cracking did not have much influence on the behaviour of the load-shortening curve, as expected. After global buckling, the axial stiffness was very small, with a very low increase in the load. The onset of collapse occurred for \( u_z = 2.42 \text{ mm} \) due to fiber kinking, arising in the blade of outer stiffener 1. This failure mechanism propagated to more FE and layers, but low loss of stiffness was also predicted from that point on.

On the other hand, panel design T6 was naturally predicted to be the stiffest of all variations of the T-shaped panels, with the highest value of \( P_u = 141.8 \text{ kN} \) and the highest global buckling load \( (P_{cr,global} = 107.4 \text{ kN}) \). The stiffness after global buckling was slightly higher and the \( \Delta u_{gc} \) slightly lower compared than those of panel T5, but both were much higher than the values exhibited by T4 design. Nevertheless, the predicted value of \( \Delta u_{lg} \) was the smallest of the T-shaped variations, which means that stringers buckled shortly after the onset of local buckling. This panel also displayed different composite damage mechanisms, and its collapse occurred for an axial shortening \( u_z = 2.49 \text{ mm} \) due to fiber kinking, which was induced by an abrupt change in the deformation pattern (sudden inward displacement of two inner stiffeners). Right after the collapse point this panel exhibited a
stronger reduction of the load-carrying capacity than panel T5, where the load and axial stiffness decreased less sharply. Summing up, panel design T6 presented the highest value of $P_u$ combined with almost the same value of $\Delta u_{gc}$ as that of T5 panel, but presented the lowest value of $\Delta u_{lg}$ and collapsed with a “catastrophically” loss of load carrying capacity, whereas the latter collapsed gradually in a “smoother” way.

Panel design I was the stiffest of all panel designs, since it is the one with largest total stiffener cross-section area. This panel was capable of withstanding a maximum axial load $P_u = 217.8$ kN, which is almost two times that of panel T5. With a considerably value of $\Delta u_{lg}$, the panel only displayed small local buckles between the stiffeners, with 15 buckling waves per stiffener accompanied by matrix damage mechanisms up to the global buckling point. However, this panel collapsed suddenly for $u_z = 2.18$ mm, with a strong reduction of the load-carrying capacity of about $\Delta P = 44.3$ kN, which can be perceived from the substantial negative slope of the load-shortening curve from $u_z = 2.18$ mm to $u_z = 2.32$ mm. This first loss of stiffness corresponded to the onset of global buckling, where the left outer stiffener (stringer 1) buckled outwards (away from the centre of curvature). Hence, this panel design was considered to have a null $\Delta u_{gc}$, and thus was the most brittle of all panels. For a shortening $u_z = 2.20$ mm another global buckling shape developed at the right outer stiffener (stringer 5), which buckled away from the centre of curvature and warped some FE located at centre of the top flange of the stiffener. That significant twisting movement generated promptly fiber kinking at those FE and caused the second loss of stiffness between $u_z = 2.50$ mm and $u_z = 2.75$ mm.

Panel design C presented a similar behavior as that of panel T5, with practically the same initial stiffness, since both panels possess the same total stiffener cross-section area. The value of $P_u$ was slightly higher in the former compared to the latter ($P_u = 118.9$ kN (design C) rather than $P_u = 110.7$ kN (design T5)) but the shortening at which collapse occurred ($u_c = 2.50$ mm) was the same in both
panels. Even though these two panels had the same amount of CFRP material, one half of each stringer flange of panel T5 was removed and placed on the top of the stringer blades to form panel C. On the other hand, the amount of adhesive material used in the latter was half of that used in the former. Though the global buckling of panel C occurred at higher values of the axial load and shortening, thus displaying a higher value of $\Delta u_{lg}$, $\Delta u_{gc}$ was not as “stable” as the one exhibited by panel design T5, as the first displayed approximately zero stiffness from $u_z = 1.67$ mm to $u_z = 2.11$ mm and a load decrease from $u_z = 2.11$ mm to $u_z = 2.31$ mm. In contrast, panel T5 presented a very “stable” $\Delta u_{gc}$ with a considerable constant stiffness up to the collapse point.

Panel design J presented a similar initial stiffness as that of T6 panel up to the first global buckling point, which occurred for a shortening $u_z = 1.79$ mm, corresponding to the first loss of stiffness, generated by an outward buckle of one of the outer stiffeners. The maximum load was $P_u = 156.3$ kN and was achieved right before global buckling, which was immediately followed by a considerable loss of the axial stiffness, as well as by the development of a second global buckling shape for $u_z = 1.87$ mm. This latter buckling shape involved another outward buckle of the opposite outer stiffener, as well as an inward displacement of stiffeners 2 and 3, which caused fiber kinking in stiffeners 2 and 4 and, consequently, the collapse of the panel for $u_z = 1.99$ mm. This panel exhibited a considerable value of $\Delta u_{lg}$, but displayed a low value $\Delta u_{gc}$ because shortening at collapse and shortening at the first global buckling were close to each other ($\Delta u_{gc} = 0.2$ mm).

Panel design $\Omega$ was the second stiffest among all panels and it presented the second highest value of $P_u$. This panel exhibited a very similar behavior as that of the panel J up to the first global buckling point, with the same initial stiffness (both presenting $\frac{dP}{du_z} \approx 92$ kN/mm), which is, likewise, identical to that of panel T6. Moreover, panels $\Omega$ and J present the same global buckling load. Panel $\Omega$ displayed an identical value of $\Delta u_{lg}$ as that of design I, but presented a much higher $\Delta u_{gc}$ ($\Delta u_{gc} = 0.84$ mm), whereas panel design I collapsed right after global buckling. This panel design ($\Omega$) collapsed for a shortening of about $u_z = 2.58$ mm, due to a sudden and strong mode switch involving stringers 4 and 5, which generated fiber kinking in the inner stiffener 4, specifically at the 0º plies of the top flange and at the 45º and -45º plies of the lateral blades. Finally, the second abrupt load decrease for $u_z = 3.50$ mm was caused by breakage of the fibers at the topmost ply of the skin in the vicinity the second stiffener.

The deformation patterns of all panel designs in compression are depicted in Table C-2 (Appendix-C) for the three load levels (local buckling, global buckling and collapse).

Based on some of the conclusions achieved by the previous comparisons, an attempt was made for improving the structural behavior of panel design $\Omega$, specifically with the objective of enhancing its axial stiffness after global buckling, as it was perceived that this panel had the most noteworthy results. It exhibited the second highest value of $P_u$, the highest value of $\Delta u_{lg}$ (similar to the I design) and a significant value of $\Delta u_{gc}$, as well as the largest axial shortening at collapse. Thus, it was found that if the laminate layups of the three flanges of each stiffener were inverted (from [(45, -45)s, 0]s to
[0\text{\,}(45, -45)\text{\,}3\text{\,}]$, the structural response of the panel would become even better, with the stiffness after global buckling being substantially increased (from $\frac{\Delta P}{\Delta u_g} \approx 8.3 \text{ kN/mm}$ to $\frac{\Delta P}{\Delta u_g} \approx 23.7 \text{ kN/mm}$), as it can be observed in Table 13 (panel $\Omega$-modified) and Figure 51. This resulted in higher values of both $P_u$ and $\Delta u_{gc}$, with increases of 13.3% and 22.5%, respectively. This modification was also implemented to the other panel designs, but the numerical results did not suffer significant variations, as depicted in Appendix C.

The current industrial design scenario composite stiffened structures, depicted in Figure 52 (a) (simplified representation), shows that the design limit load is selected taking into consideration that damage is, so far, not allowed in any flight condition. The ultimate load is typically 150% of the design limit load to ensure a margin of safety. However, with this design scenario there is a large unemployed structural reserve capacity between the ultimate load and collapse. In future designs (Figure 52 (b)) the onset of damage is allowed in the safety region, the limit load is much larger than the first local buckling load and the ultimate load is shifted towards the structural collapse as close as possible.

The assessment and comparison of the achieved numerical results is believed to offer a direct contribution to the future designs of composite stiffened structures. The best panel design was selected as the one that could withstand the axial load and, at the same time, be the one with the highest structural reserve capacity between the first buckling load and collapse (i.e., with the highest $\Delta u_{lg}$ and $\Delta u_{gc}$). Following this, panel design $\Omega$-modified was chosen as the best design among all configurations. This panel exhibits a progressive change from local to global buckling, a large and “stable” postbuckling shortening from 0.57 mm to 2.90 mm of axial shortening and a value of $P_u = 178.5 \text{ kN}$. Future experimental and numerical investigations are also recommended in order to incorporate the effects of delamination and skin-stringer debonding in this panel design, which will lead to more accurate predictions of structural collapse. Additionally, this panel design is lighter than panel T5 studied in the COCOMAT project, as it required approximately 524 690 mm$^3$ of CFRP prepeg IM7/8552 UD, whereas T5 panel needed about 787 800 mm$^3$ of the same carbon/epoxy composite material, which goes along with the continuous demands for decreases in the structural weight.
4.4. Postbuckling of panels under bending

The analysis of the panels subjected to bending was also incorporated in this work because it is known that the axial stresses developed along the circumferential direction of a fuselage are not uniform and may vary linearly, and thus the stress gradient arising from these 2nd order forces is well defined through a linear stress distribution equivalent to the application of a bending moment. On the other hand, studies and numerical analysis on this load case were not performed in the COCOMAT project. Therefore, accurate predictions of damage and collapse regarding this load case are worthy of being studied and can contribute to the design of more efficient composite structures. Once again, due to the lack of experimental data regarding this load case and for the purpose of comparison of the different panel designs, the analysis of the panels under bending was only made considering damage of the composite material (DM-H).

4.4.1. Reference panel T5

Figure 53 shows the bending moment-rotation curve of the panel design T5. The prescribed rotation $|\theta_y| = 0.015 \text{ rad}$ around y-axis was chosen in order to yield, in the first increment, an equivalent axial displacement, as that of the compressive load case, on the outer nodes of the loaded edge (where $u_z$ is maximum). The bending moment ($M$) was found by requesting and summing the values of the reaction moment in the y-direction (RM-2 in Abaqus) for the whole set of nodes with all DOF restrained (located at the fixed/clamped side of the panel). The rotations ($\theta_y$) were given directly by that of the reference point (RP). In the whole panel, rotation around y-axis was permitted but the out-of-plane displacements (along y-axis) were not permitted in the potting region.

![Bending moment-rotation curve of the panel design T5](image)

The results presented the typical behavior of stiffened panels subjected to this loading mode and suggested the presence of two main characteristic bending moment levels. Again, the structure subjected to bending developed specific mode shapes in order to minimize the potential energy. Figure 54 depicts the axial displacements ($u_z$) of the initial shape (undeformed) and the deformed

Figure 53: Bending moment-rotation curve of the panel design T5
shape of the panel after collapse. It is clear, by examining the axial displacements of the outmost
nodes of the loaded edge, that one side of the panel was subjected to compressive (in the left side of
Figure 54) and the other (in the right side of Figure 54) to tensile stresses. The first loss of stiffness
(point A in Figure 53) was reached for a rotation $\theta_y = 5.0 \times 10^{-3}$ rad and was associated to the out-of-
plane displacements of the left outer stiffener (stiffener 5), which buckled inwards (towards the centre
of curvature) due to the high compressive stresses verified in that side of the panel. That point of
rotation corresponded to the first “knee” in the curve presented in Figure 53, where the flexural
stiffness slightly decreased (from $\frac{\Delta M}{\Delta \theta_y} \approx 2.752 \text{kNm/rad}$ to $\frac{\Delta M}{\Delta \theta_y} \approx 0.85 \text{kNm/rad}$).

From a rotation of circa $\theta_y = 5.9 \times 10^{-3}$ rad, the side of the panel in compression rapidly developed ply
damage in the form of matrix crushing at the outmost layers of the skin and stringer blade and flange
(bottom and top plies), as indicated by the unit value of the damage initiation variable $HSNMCCRT$
(Figure 55). For that value of rotation, the top ply of the skin oriented at 90º with z-axis had the highest
number of damaged elements (quantified by the output variable $DAMAGEMC$), as depicted in
Appendix-C.

The collapse of the panel (point B) occurred for $\theta_y = 7.92 \times 10^{-3}$ rad with a sharp reduction of the
bending moment-carrying capacity and flexural stiffness. It was caused by fiber kinking in several FE
at the outer stringer blade in the side of the panel in compression. The damaged FE varied accordingly
with the ply orientation, as shown in Figure 56. After collapse, the curve exhibited little periodic ups
and downs, but the bending moment was nearly constant up to the final rotation value of
$\theta_y = 15.0 \times 10^{-3}$ rad.

Figure 54: Initial shape/ undeformed (grey color) and the deformed shape of the panel after collapse
Figure 55: Damage initiation (matrix crushing mode) for multiple section points for $\theta_y = 5.9 \times 10^{-3}$ rad

(a)  
(b)  
(c)  

Figure 56: Degree of damage (fiber kinking mode) of (a) the $+45^\circ$, (b) the $-45^\circ$ and (c) $0^\circ$ plies of the outer left stringer blade for $\theta_y = 7.92 \times 10^{-3}$ rad

4.4.2. Influence of stringer number and shape

In this sub-section, the analysis of the panels subjected to bending was extended to the additional seven panel designs (T4, T6, I, J, $\Omega$ and $\Omega$-modified). The same rotation of $|\theta_y| = 0.015$ rad with respect to the y-axis was applied to the RP of all panel designs to assess and compare their structural behaviour.

Table 14 presents the main results of the eight panels analyzed (including panel $\Omega$-modified), specifically (i) the rotation and bending moment associated to the first loss of flexural stiffness (FLFS), caused by inward buckling of the outer stiffeners in the region of the panel subjected to compression, (ii) $M_u$, which is the maximum bending moment with respect to the y-axis that the panel can withstand, (iii) the rotation at collapse ($\theta_c$), and (iv) the postbending rotation ($\Delta \theta_y$), defined as the rotation between the FLFS and collapse.
Table 14: Comparison of results between the 7 panel designs (bending)

<table>
<thead>
<tr>
<th>Panel designs</th>
<th>FLFS</th>
<th>M (kNm)</th>
<th>Mu (kNm)</th>
<th>( \theta_{yc} ) (mm)</th>
<th>( \Delta \theta_g ) (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T5</td>
<td>5.0 x10^{-3}</td>
<td>12.8</td>
<td>15.2</td>
<td>7.9 x10^{-3}</td>
<td>2.9 x10^{-3}</td>
</tr>
<tr>
<td>T4</td>
<td>4.9 x10^{-3}</td>
<td>11.2</td>
<td>13.3</td>
<td>8.2 x10^{-3}</td>
<td>3.3 x10^{-3}</td>
</tr>
<tr>
<td>T6</td>
<td>5.4 x10^{-3}</td>
<td>15.3</td>
<td>17.2</td>
<td>8.2 x10^{-3}</td>
<td>2.8 x10^{-3}</td>
</tr>
<tr>
<td>I</td>
<td>9.2 x10^{-3}</td>
<td>30.8</td>
<td>30.8</td>
<td>9.2 x10^{-3}</td>
<td>0.0</td>
</tr>
<tr>
<td>C</td>
<td>7.6 x10^{-3}</td>
<td>18.3</td>
<td>19.8</td>
<td>9.8 x10^{-3}</td>
<td>2.2 x10^{-3}</td>
</tr>
<tr>
<td>J</td>
<td>8.2 x10^{-3}</td>
<td>23.4</td>
<td>23.3</td>
<td>9.1 x10^{-3}</td>
<td>0.9 x10^{-3}</td>
</tr>
<tr>
<td>Ω</td>
<td>7.4 x10^{-3}</td>
<td>21.5</td>
<td>25.2</td>
<td>10.6 x10^{-3}</td>
<td>3.2 x10^{-3}</td>
</tr>
<tr>
<td>Ω-modified</td>
<td>7.4 x10^{-3}</td>
<td>21.5</td>
<td>25.9</td>
<td>11.2 x10^{-3}</td>
<td>3.8 x10^{-3}</td>
</tr>
</tbody>
</table>

The assessment of the bending moment-rotation curves (Figure 57) of the eight panel designs presented a close correlation with those of the load-axial shortening curves of the same panels subjected to axial compression. The side of the panels in compression developed identical ply damage mechanisms, though the affected FE evidently differed. The collapse of the panels occurred with a sharp reduction of the bending moment-carrying capacity and flexural stiffness, and was once more induced by the appearing of fiber kinking in several FE in the outer stringers.

Panel design T4 was again the one displaying the lowest stiffness of the three T-shaped versions as well as the lowest value of \( M_u = 13.3 \) kNm, which did not occur at the point of global buckling as in the previous, compressive, postbuckling analysis, but at the collapse point, which took place for \( \theta_{yc} = 8.2 \times 10^{-3} \) rad. Nevertheless, this panel exhibited a slightly higher postbending rotation than the other T-shaped panels. On the other hand, panel design T6 was again predicted to be the stiffest of all T-shaped variations, with the highest initial flexural stiffness, the highest value of \( M_u = 17.2 \) kNm and a slightly lower postbending rotation in comparison with that of the other panels. This panel collapsed for \( \theta_{yc} = 8.2 \times 10^{-3} \) rad and after collapse the bending moment was nearly constant up to the final value of rotation.

Panel design I displayed the highest flexural stiffness among all panel designs, with the panel being capable of withstanding a maximum bending moment of \( M_u = 30.8 \) kNm, which is twice that of panel T5. This panel collapsed for \( \theta_{yc} = 8.2 \times 10^{-3} \) rad, with a reduction of \( \Delta M = 4.95 \) kNm in the load-carrying capacity. It was considered that this panel design had a postbending rotation of \( \Delta \theta_g = 0.0 \) rad, since the flexural stiffness was constant up to the point of collapse and thus the most brittle behavior, among all designs, was observed.
Figure 57: Bending moment-rotation curve of eight panel designs under bending

Panel design C displayed an identical initial flexural stiffness as that of panel T5 up to $\theta_y = 5.0 \times 10^{-3}$ rad, though from this point on it no longer showed a structural behaviour analogous to its behaviour under axial compression, as it occurred with panel T5. Collapse occurred for $\theta_{yc} = 9.85 \times 10^{-3}$ rad, though the largest moment reduction arose at $\theta_y = 12.31 \times 10^{-3}$ rad and was generated by a significant increase of the inward out-of-plane displacements in the side of the panel in compression, which likewise caused fiber kinking in the closest inner stiffener. This panel attained a maximum moment $M_u = 19.8$ kNm, higher than any of the T-shaped configurations.

Panel design J presented an identical initial stiffness as that of the panel $\Omega$ up to the FLFS (equally confirmed in the postbuckling analysis of both panels in compression). The collapse of the panel occurred for $\theta_{yc} = 9.0 \times 10^{-3}$ rad and the maximum moment was $M_u = 23.3$ kNm, achieved at the FLFS point, which was followed by a constant value of the moment (zero stiffness). A second and more significant moment reduction arose for $\theta_y = 11.18 \times 10^{-3}$ rad and was again caused by a strong and sudden increase of the inward out-of-plane displacements in the outer stiffener. This panel exhibited a very short postbending rotation of just $\Delta \theta_g = 0.9 \times 10^{-3}$ rad.

Panel design $\Omega$ was the second stiffest among all panels, as it presented the second highest value of $M_u$ ($M_u = 25.2$ kNm). This panel collapsed for a rotation of about $\theta_{yc} = 10.6 \times 10^{-3}$ rad and exhibited a considerable post bending rotation of $\Delta \theta_g = 3.2 \times 10^{-3}$ rad. On the other hand, the attempt of improving its structural response by inverting the laminate layups of the three flanges of each stiffener was also made. However, the curves indicated that the structural behavior of the new panel $\Omega$-modified subjected to this loading mode was very similar to that of panel $\Omega$, with just a slight increase of the maximum attained moment ($M_u = 25.9$ kNm) as well as of the rotation at collapse, which yielded a larger post bending rotation ($\Delta \theta_g = 3.8 \times 10^{-3}$ rad).
Panel Ω-modified was considered to have produced the most noteworthy results regarding the bending response to a prescribed rotation around y-axis, as it exhibited the second highest value of $M_u$, combined with a significant post bending rotation, as well as the largest rotation at collapse. The panel design 1 also presented distinguished results with this loading case, though it started to collapse right after the FLFS point, which does not go along with the future aims for the design of stiffened composite panels, where the stiffeners can withstand significant deformations in the safety region before the ultimate loads and collapse.
5. Conclusions and Future Developments

5.1. Conclusions

The work presented in this thesis was mainly focused on three objectives. The first was to integrate different modeling approaches in the FEA to represent the critical damage mechanisms in a thin-walled stiffened CFRP panel under axial compression, comprising T-shaped stringers similar to those of the panel studied in the COCOMAT project. Three damage models were implemented and evaluated and then considered as an alternative to the user subroutines previously developed in the that project, which are very time-consuming when used with ABAQUS. With the adopted approaches, the prediction of more realistic deformation patterns and closer approximations between numerical and experimental load-axial shortening curves were attempted, but the evaluation of the different failure criteria on the structural behavior of the panel was the first principal purpose.

The second main focus of this work was to create several panels designs with different stringer geometries to evaluate their postbuckling structural behavior under axial compression. The load-carrying capacity and collapse of those panels were analyzed and compared. Finally, the last goal of this work will be to incorporate an additional bending analysis to all panel configurations, as the study of this load case can contribute to the design of more efficient composite structures.

A wide range literature review was presented in Chapter 2, in which the main aspects relevant to the analysis of composite stiffened structures and modelling of damage up to collapse were covered. Three important conclusions were drawn from this chapter. The first was that the concept of postbuckling design has the potential to improve the structural efficiency. The second was that the behavior of thin-walled composite structures in compression is a highly nonlinear event, comprising material and geometric non-linearities, and thus fully nonlinear analyses are necessary to adequately capture the structural behavior of the panels. The last main conclusion was that the numerical model developed in the COCOMAT project, also presented in Chapter 2, was not able to capture the deformations patterns of the panel observed experimentally. Additionally, the numerical results attained almost completely misrepresented the degree of damage of the adhesive layer. Nevertheless, this project has shown that the incorporation of damage into the FE models is essential.

Regarding the postbuckling analysis of panel T5 (panel with five T-shaped stringers) under axial compression, which was created based on the COCOMAT design (D1), it was clear that an analysis approach combining more than one damage model would be highly attractive, as it would allow the potential interactions between strength-based failure criteria and fracture mechanics to be investigated. The numerical results concerning this panel design were then compared with the experimental data, and it was shown that the approach with DM-HX (Hashin + XFEM) led to the closest prediction of the panel behavior in terms of the load-shortening curves and of the shortening and load for which collapse occurred. On the other hand, the model with DM-HC (Hashin + cohesive elements) was able to represent an asymmetric deformation pattern close to that observed experimentally as well as a good prediction of the areas where skin-stringer debonding took place.
The postbuckling analysis of the different panels in compression and bending permitted to identify their load-shortening and moment-rotation response to an applied axial displacement and rotation, respectively, the onset of damage mechanisms and their capability to resist further increase in load after the first global buckling load. Panel design Ω-modified was chosen as the best design in terms of structural efficiency as it evidenced the highest exploitation of postbuckling reserve strength. This panel is also lightest of the ones studied, in particular, lighter than the panel studied in the COCOMAT project, and thus it is recommended to be studied for possible future applications.

Hence, the achieved numerical results are believed to offer a direct contribution to the future designs of composite stiffened structures, as well as to be a contribution to the aim of structural weight reduction, and consequently to allow the European aircraft industry to reduce development and operation costs in the short and long term. Although this project was mainly focused on fuselage panels under axial compression and bending, the analysis approach and the damage models applied can be easily transferable to other composite structures and loading cases.

5.2. Future developments

Besides the further experimental and numerical investigations recommended in section 4.3, for more accurate predictions of structural collapse of the most efficient panel design (Ω-modified), studies with panels subjected to dynamic loading are also recommended. Presently, the static (or quasi-static) load-carrying capacity is the most used approach for the design process of composite stiffened structures. However, dynamic loading may lead to substantially lower buckling loads, especially when a short duration axial load is rapidly applied and then held fixed (e.g. landing impact of an aircraft or during gust loading) [29]. Therefore, the reduction of the load-carrying capacity of shell structures under dynamic loading must be taken into consideration in future designs regarding safety, and accurate experimental and numerical investigations concerning this critical load case are recommended.
6. References


Appendix A

Literature Review

Figures A-1 to A-5 illustrate the principal damage mechanisms in composite structures.

Figure A-1: Fiber fracture/rupture

Figure A-2: Fiber kinking/micro-buckling

Figure A-3: Matrix crushing

Figure A-4: Matrix cracking
**Hashin Damage Output Variables:**

In Abaqus, the damage initiation output variables associated with each initiation criterion (indexes $F_f^t$, $F_f^c$, $F_m^t$ and $F_m^c$), as well as the variables specifically related to damage evolution in fiber-reinforced composites are the following [4]:

<table>
<thead>
<tr>
<th>Output Variables</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>HSNFTCRT</td>
<td>maximum value of the fiber tensile initiation criterion experienced during the analysis</td>
</tr>
<tr>
<td>HSNFCCRT</td>
<td>maximum value of the fiber compressive initiation criterion experienced during the analysis</td>
</tr>
<tr>
<td>HSNMTCRT</td>
<td>maximum value of the matrix tensile initiation criterion experienced during the analysis</td>
</tr>
<tr>
<td>HSNMCCRT</td>
<td>maximum value of the matrix compressive initiation criterion experienced during the analysis</td>
</tr>
<tr>
<td>DAMAGEFT</td>
<td>Fiber tension damage variable</td>
</tr>
<tr>
<td>DAMAGEFC</td>
<td>Fiber compression damage variable</td>
</tr>
<tr>
<td>DAMAGEMT</td>
<td>Matrix tension damage variable</td>
</tr>
<tr>
<td>DAMAGEMC</td>
<td>Matrix compression damage variable</td>
</tr>
<tr>
<td>DAMAGEGHR</td>
<td>Shear damage variable</td>
</tr>
<tr>
<td>STATUS</td>
<td>The status of an element is 1.0 if the element is active and 0.0 otherwise. It only reaches the value of 0.0 if damage has occurred in all the damage modes</td>
</tr>
</tbody>
</table>
Figure A-6 illustrates the application of the VCCT method for 8-node solid elements (for calculating the strain energy release rates at the crack tip).

The output variables related to cohesive elements with traction-separation response are the following [4]:

Table A-2: output variables related to cohesive elements

<table>
<thead>
<tr>
<th>Output Variables</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>STATUS</td>
<td>The status of an element is 1.0 if the element is active and 0.0 otherwise</td>
</tr>
<tr>
<td>MAXSCRT</td>
<td>Maximum value of the nominal stress damage initiation criterion at a material point during the analysis</td>
</tr>
<tr>
<td>QUADSCRT</td>
<td>Maximum value of the quadratic nominal stress damage initiation criterion at a material point during the analysis</td>
</tr>
<tr>
<td>SDEG</td>
<td>Overall value of the scalar damage variable, D</td>
</tr>
</tbody>
</table>
COCOMAT project

Figure A-7 illustrates the basic sizes of the P23 panel that was used in the COCOMAT experimental and simulations tests and figure A-8 depicts the tested panel in the DLR’s buckling facility.

Figure A-7: Basic sizes of the P23 panel [8]

<table>
<thead>
<tr>
<th>Basic sizes:</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius [mm]</td>
<td>R</td>
</tr>
<tr>
<td>Arc length [mm]</td>
<td>2*b₂+c</td>
</tr>
<tr>
<td>Length of the panel</td>
<td>H</td>
</tr>
<tr>
<td>Free length [mm]</td>
<td>h</td>
</tr>
<tr>
<td>Stringer distance</td>
<td>2*b₁</td>
</tr>
<tr>
<td>Stringer distance</td>
<td>2*b₂</td>
</tr>
<tr>
<td>Width of stringer</td>
<td>f</td>
</tr>
<tr>
<td>Height of stringer web</td>
<td>h₃</td>
</tr>
</tbody>
</table>

Figure A-8: Panel in the buckling facility of DLR [8]
Figure A-9 gives the load-shortening curve of the numerical analysis in Abaqus without inclusion of degradation and the experimental curve for comparison.

![Load-shortening curve](image)

*Figure A-9: Load-shortening curve of the analysis without degradation (blue) and the experimental test (green) [8]*

Figure A-10 depicts the damage observed in the adhesive layer and the subsequent areas of skin-stiffener debonds.

![Damage observation](image)

*Figure A-10: Visualizing damage in the adhesive and skin-stringer separation: using ultrasonic flaw echo (left); and thermographic (right)*
Appendix B

FE Model Description

Figure B-1 illustrates the 50 partitions created to improve the accuracy of the DM-HX model. Figure B-2 depicts the final mesh of the panel design T5 with DM-H applied. The second mesh developed, which incorporates damage in the adhesive, is similar but with 10 elements in the stringer flange instead of 6 (as well as in the same regions of the skin and adhesive as matched meshes were used).

![Figure B-1: partitions in the adhesive layer](image1)

![Figure B-2: final mesh of the panel design T5](image2)

Table B-1 indicates the mesh densities of all panel designs. Figure B-3 shows the rigid body definition and the location of the RP in the assembly.
Table B-1: mesh densities of all panel designs and respective illustration

<table>
<thead>
<tr>
<th>Model</th>
<th>Number of elements</th>
<th>Illustration</th>
</tr>
</thead>
<tbody>
<tr>
<td>T4</td>
<td>9 828</td>
<td>![Illustration of T4]</td>
</tr>
<tr>
<td>T6</td>
<td>12 636</td>
<td>![Illustration of T6]</td>
</tr>
<tr>
<td>I</td>
<td>13 650</td>
<td>![Illustration of I]</td>
</tr>
<tr>
<td>C</td>
<td>9 906</td>
<td>![Illustration of C]</td>
</tr>
<tr>
<td>Model</td>
<td>Number of elements</td>
<td>Illustration</td>
</tr>
<tr>
<td>-------</td>
<td>--------------------</td>
<td>--------------</td>
</tr>
<tr>
<td>$J$</td>
<td>12,480</td>
<td><img src="image1.png" alt="Illustration" /></td>
</tr>
<tr>
<td>$\Omega$</td>
<td>13,260</td>
<td><img src="image2.png" alt="Illustration" /></td>
</tr>
</tbody>
</table>

*Figure B-3: rigid body definition and the location of the RP in the assembly*
### Numerical Simulations and Results

#### Linear Results (First buckling mode shape):

*Table C-1: First eigen shapes of the several panel designs*

<table>
<thead>
<tr>
<th>Panel Design</th>
<th>First eigen mode (scale factor=20)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td><img src="image1.png" alt="T1" /></td>
</tr>
<tr>
<td>T2</td>
<td><img src="image2.png" alt="T2" /></td>
</tr>
<tr>
<td>T3</td>
<td><img src="image3.png" alt="T3" /></td>
</tr>
<tr>
<td>I</td>
<td><img src="image4.png" alt="I" /></td>
</tr>
<tr>
<td>Panel Design</td>
<td>First eigen mode (scale factor=20)</td>
</tr>
<tr>
<td>--------------</td>
<td>-----------------------------------</td>
</tr>
<tr>
<td>C</td>
<td>![Image of panel C]</td>
</tr>
<tr>
<td>J</td>
<td>![Image of panel J]</td>
</tr>
<tr>
<td>Ω</td>
<td>![Image of panel Ω]</td>
</tr>
</tbody>
</table>

Numbering of the stringers:

![Stringer numbering diagram]

*Figure C-1: stringer numbering*
DM-H (Hashin):

Figure C-2 illustrates the fiber-matrix shear mode (given by the output variable DAMAGESHR) for the bottom and top plies of the skin at an axial shortening of $u_z = 1.09$ mm.

Figure C-2: Degree of damage (fiber-matrix shear mode) for the bottom (left) and top (right) plies of the skin for $u_z = 1.09$ mm

Figure C-3 depicts the matrix cracking failure mode after collapse for multiple section points.

Figure C-3: Degree of damage (matrix cracking mode) for $u_z = 3.3$ mm
Figure C-4 illustrates the fibre kinking failure mode for one ply of the stringer blade oriented at 0º (bottom) and at 45º (top) at $u_z = 3.3$ mm (beyond collapse).

Figure C-5 illustrates the fiber kinking damage mode across the stringer blade 2, close to the potting region on the non-loading side, induced by the skin-stringer separation captured by the model with DM-HC.

DM-HC:

Figure C-5: Fiber kinking in stringer blade 2 for $u_z = 2.36$ mm
DM-HX (Hashin+ XFEM):

Figure C-6 depicts the second global buckling shape developed in the vicinity of two of the inner stiffeners (stringers 3 and 4).

Figure C-6: Second global buckling shape developed in the vicinity of two of stringers 3 and 4 from $u_z = 1.95 \text{ mm}$ to $u_z = 2.05 \text{ mm}$

Figure C-7 shows the FE affected by matrix crushing and cracking for a value of shortening beyond collapse.

Figure C-7: Matrix crushing (left) and cracking(right) at the bottom ply of skin and stringers after collapse
Postbuckling analysis of panels in compression (additional panel designs):

Table C-2: Deformation patterns of all panel designs. The red/yellow colours represent inward displacement and the blue/green outward displacement.

<table>
<thead>
<tr>
<th>Panel designs</th>
<th>Illustration (out-of-plane displacements ($u_y$))</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Local buckling</td>
</tr>
<tr>
<td>T5</td>
<td><img src="image1" alt="Local buckling" /></td>
</tr>
<tr>
<td>T4</td>
<td><img src="image4" alt="Local buckling" /></td>
</tr>
<tr>
<td>T6</td>
<td><img src="image7" alt="Local buckling" /></td>
</tr>
<tr>
<td>Panel designs</td>
<td>Illustration (out-of-plane displacements ($u_y$))</td>
</tr>
<tr>
<td>---------------</td>
<td>-----------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>Local buckling</td>
</tr>
<tr>
<td>I</td>
<td><img src="image1.png" alt="Local buckling Image" /></td>
</tr>
<tr>
<td>C</td>
<td><img src="image1.png" alt="Local buckling Image" /></td>
</tr>
<tr>
<td>J</td>
<td><img src="image1.png" alt="Local buckling Image" /></td>
</tr>
</tbody>
</table>
### Illustration (out-of-plane displacements ($u_y$))

<table>
<thead>
<tr>
<th>Panel designs</th>
<th>Local buckling</th>
<th>Global buckling</th>
<th>Collapse</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Omega$</td>
<td><img src="image1" alt="Local buckling" /></td>
<td><img src="image2" alt="Global buckling" /></td>
<td><img src="image3" alt="Collapse" /></td>
</tr>
<tr>
<td>$\Omega$-modifié</td>
<td><img src="image4" alt="Local buckling" /></td>
<td><img src="image5" alt="Global buckling" /></td>
<td><img src="image6" alt="Collapse" /></td>
</tr>
</tbody>
</table>
Modification of the laminate layups of the flanges (panel designs T5, T4, T6, I, C and J)

**Figure C-8:** Load-shortening curve of two versions of panel design T5

**Figure C-9:** Load-shortening curve of two versions of panel design T4
Figure C-10: Load-shortening curve of two versions of panel design T6

Figure C-11: Load-shortening curve of two versions of panel design I

Figure C-12: Load-shortening curve of two versions of panel design C
Figure C-13: Load-shortening curve of two versions of panel design J

Bending analysis:

Figure C-14: Degree of damage (matrix crushing mode) for the top ply of the skin for $\theta_y = 5.86 \times 10^{-3}$ rad