A NUMERICAL APPROACH FOR THE SHAPE OPTIMIZATION OF
WOVEN FABRIC COMPOSITE STRUCTURAL ELEMENTS

Antonios St. Koumpias

Laboratory of Technology & Strength of Materials

Supervisor: Prof. Spiros G. Pantelakis
"A numerical approach for the shape optimization of woven fabric composite structural elements"
Given this opportunity I would like to express my gratitude and respect to all the people that made this work possible, beginning with my supervisor Prof. Pantelakis for giving me the opportunity to work in LTSM. However, most importantly he believed in my potential even when I did not believe in myself and he was more than understanding during the hard times. I owe him a lot and I hope that I have fulfilled his expectations. In a few words he played a major role in what I am today.

Alongside Prof. Pantelakis, I would like to express my deepest appreciation to Assistant Prof. Konstantinos I. Tserpes for his excellent guidance during my work at LTSM. This work would have been almost impossible without his support and constructive criticism. He always provided me with valuable advices and indicated me how to work in a correct and efficient manner. Looking back, I consider myself very lucky that I had the opportunity to collaborate with him.

I would also like to thank Associate Prof. Georgios Labeas and Emeritus Prof. Theodoros Kermanidis. They were more than willing to answer my questions and offer advices and new ideas whenever weeded.

This section wouldn’t be complete without mentioning the great help offered by Dr. Ioannis Diamantakos who gave a solution in every problem in the field of finite element modeling.

Additionally, I gratefully acknowledge the help, of Prof. Spilios D. Fassois who not only helped me with the corrections of this work but also offered me his advices on all technical matters

Special thanks to my colleagues in the LTSM lab for all the good and the bad times we had together! Firstly my officemates, Nikos Perogamvros, Rita Setsika and Giannis Floros who tolerated me these years! Also Akis Markatos, Apostolos Hamos, Michalis Papadopoulos, Georgios Diamantogiannis, Xristos Katsiropolous, Vassilis Passialis, Angeliki Chanteli, Antonis
Stamopoulos, Kostas Fotopoulos, Vaggelis Ptochos, Georgios Milonas and Victor Bulungu
Watiti thank you all!

Of course, I would like to thank my parents for their emotional support and guidance. Without their help, patience and understanding I wouldn’t be in this position today.

Last but not least I would like to thank my friend Kostas Chronopoulos and my girl Ellie Fassoi who have always been there for me.
# Table of Contents

**Acknowledgements** ......................................................................................... i

**Table of Contents** .................................................................................................. iii

**List of Figures** ........................................................................................................ vi

**List of Tables** .......................................................................................................... xi

**List of Abbreviations** ............................................................................................ xii

**Abstract** .................................................................................................................. xiii

**Περίληψη** ................................................................................................................ xiv

1. **Subject of the thesis** .......................................................................................... 1  
   1.1. The Technological problem ............................................................................. 1  
   1.2. Aim and Approach ......................................................................................... 4  
   1.3. Thesis Summary ............................................................................................. 5  

2. **Literature review** ................................................................................................ 8  
   2.1. Optimization of composite structures ............................................................. 8  
   2.2. Optimization of bonded joints ....................................................................... 12  
   2.3. Joining Elements ............................................................................................. 17  
   2.4. Woven Composite materials .......................................................................... 21  
      2.4.1. NCF Composite materials ....................................................................... 22  
      2.4.2. 3D FIW composite materials .................................................................. 25  

3. **Definition of Basic Terms** ................................................................................ 29  

4. **The numerical approach for the optimization of composite structures** .......... 31  
   4.1. Description of the numerical approach ............................................................ 31  
      4.1.1. PDM ........................................................................................................... 34  
      4.1.2. Optimization Module ............................................................................... 36  
         4.1.2.1. Optimization Variables ....................................................................... 36  
         4.1.2.2. Optimal Design Selection .................................................................. 37  
   4.2. Innovations ....................................................................................................... 40  

5. **Application of the numerical approach to the H-shaped joining element** ....... 41
## Table of Contents

5.1. Geometry and materials ............................................................................. 41
5.2. FE model of the joint .................................................................................. 44
5.3. Failure analysis and material property degradation ................................. 48
  5.3.1. Composite materials ........................................................................ 48
  5.3.2. Adhesive ....................................................................................... 49
5.4. Innovations ............................................................................................ 56

6. **Numerical characterization of woven composite materials** ....................... 57

6.1. Modelling of Woven composite materials .............................................. 57
  6.1.1. Geometrical pre-processors .............................................................. 59
6.2. NCF composite material ......................................................................... 61
  6.2.1. Geometrical Modelling .................................................................. 61
  6.2.2. FE modelling of the NCF RVE ......................................................... 65
  6.2.3. Failure analysis and material property degradation ......................... 68
    6.2.3.1. Tows ................................................................................... 68
    6.2.3.2. matrix ............................................................................... 71
  6.2.4. Boundary conditions ....................................................................... 71
  6.2.5. Results .......................................................................................... 73
    6.2.5.1. Validation ........................................................................... 73
    6.2.5.2. Stress-strain curves .............................................................. 74
    5.2.5.3. Damage maps ..................................................................... 77
6.3. 3D FIW composite material .................................................................... 84
  6.3.1. Geometrical Modelling .................................................................. 84
  6.3.2. FE modelling of the 3D FIW RVE ................................................... 90
  6.3.3. Failure analysis and material property degradation ......................... 92
    6.3.3.1. Tows ................................................................................... 92
    6.3.3.2. matrix ............................................................................... 94
  6.3.4. Results .......................................................................................... 94
    6.3.4.1. Validation ........................................................................... 94
    6.3.4.2. Stress-strain curves .............................................................. 97
    6.3.4.3. Damage maps ..................................................................... 99
6.4. Innovations ............................................................................................ 103
7. Optimization Results........................................................................................................................................ 105
   7.1. Joining element fabricated of NCF composite material................................................................. 105
       7.1.1. FE model validation- Initial PDM ....................................................................................... 105
       7.1.2. Optimization ....................................................................................................................... 108
       7.1.3. Optimal design selection .................................................................................................. 109
       7.1.4. Verification of optimization ............................................................................................. 111
   7.2. Joining element fabricated of 3D FIW composite material .................................................. 114
       7.2.1. FE model validation- Initial PDM ....................................................................................... 114
       7.2.2. Optimization ....................................................................................................................... 116
       7.2.3. Optimal design selection .................................................................................................. 116
       7.2.4. Verification of optimization ............................................................................................. 118
   7.3. Comparison between NCF and 3D FIW joining elements .................................................... 120
8. Conclusions ................................................................................................................................................. 122
9. Future work ................................................................................................................................................ 124
References ................................................................................................................................................... 125
Publications .................................................................................................................................................. 137
Figure 1: Commercial aircraft models and the percentage of used composite materials ............ 2
Figure 2: Optimization of an antenna structure a) initial design, b) optimized design [19] .......... 10
Figure 3: Flowchart of the methodology that combines FEA and special optimization algorithms proposed in [24] ........................................................................................................ 11
Figure 4: Shape of the adherents of a single-lap joint, a) initially and b) after 15 iterations in the OASIS-ALADDIN optimization program [30] ................................................................. 13
Figure 5: Flowchart of the methodology used in [32] ................................................................ 14
Figure 6: Double lap joint configuration studied in [35] [36] ...................................................... 15
Figure 7: Numerical prediction of the failure path of the adhesive for a filleted joint [38]. The crack starts at the tip of the filleted joint ......................................................................................... 16
Figure 8: Illustration of differently shaped joining elements ..................................................... 17
Figure 9: Illustration of a) the parts of a T-joint and b) optimization parameters studied in [41] ........................................................................................................................................... 18
Figure 10: Pi-joint and experimental setup used in [50] .............................................................. 20
Figure 11: Cured H-joint [54] ..................................................................................................... 20
Figure 12: Representation of 0/90 NCF preform where the stitching pattern and the different orientation of the tows can be seen ...................................................................................... 22
Figure 13: NCF material: a) biaxial and b) quadriaxial .............................................................. 23
Figure 14: Micrograph of the longitudinal cracks in an NCF cross-ply laminate [67] ............... 24
Figure 15: 3D FIW material a) actual material, b) explanation of the material’s structure, c) YZ view and d) isometric view ........................................................................................................... 26
Figure 16: Different configurations of 3D woven materials. a) Angle interlocked, b) orthogonally interlocked, c) fully interlocked. Taken from [85] ......................................................... 27
Figure 17: Flowchart of the numerical approach for the optimization ....................................... 32
Figure 18: PDM flowchart [102, 103] ........................................................................................ 34
Figure 19: Flowchart of the optimal design selection procedure ............................................. 38
Figure 20: Schematic illustration of an H-shaped joining element and indication different areas ......................................................................................................................... 41
Figure 21: Dimensions in mm of the H-shaped joining element ................................................................. 42
Figure 22: Initial dimensions of the specimen subjected to tension .......................................................... 43
Figure 23: Stacking sequence of a) NCF material and b) 3D FIW material ............................................. 44
Figure 24: Node position and stress output coordinates of SOLID 185 element type .......................... 45
Figure 25: Typical FE mesh of the FE model and identification of different parts ............................. 46
Figure 26: Mesh details of the different parts: a) H-profile, b) adhesive, c) gusset fillers and resin areas and d) insert ......................................................................................................................................................... 47
Figure 27: Boundary conditions for tension of the joining element ....................................................... 47
Figure 28: Softening Constitutive models .................................................................................................. 50
Figure 29: Softening stress-strain curve of the CDM [112] ................................................................. 52
Figure 30: DCB specimen. a) FE model, b) FE model details and c) boundary conditions ................. 54
Figure 31: Comparison of the simulated load-displacement curves ......................................................... 55
Figure 32: Comparison between experimental and numerical results .................................................... 56
Figure 33: Flowchart of the modelling methodology .................................................................................. 58
Figure 34: NCF RVE selection .................................................................................................................. 61
Figure 35: Flowchart of the geometrical modelling of the NCF material ............................................. 62
Figure 36: WiseTex input window. In the upper part the properties of the tows are defined.
In the lower part the definition of the stitching properties takes place ..................................... 62
Figure 37: a) Geometrical model of the RVE 1, b) stitching yarn, c) 0 layer, d) 90 layer ............. 64
Figure 38: a) Geometrical model of the RVE 2, b) stitching yarn, c) 45 layer, d) -45 layer ............ 65
Figure 39: FETex input window. In the upper part the periodic boundary conditions are inserted. At the bottom, the matrix material properties are defined ................................................................. 66
Figure 40: FE model of the two RVEs a) 0 layer, b) 90 layer, c) 45 layer .................................. 67
Figure 41: Stress–strain curves of the multi-linear isotropic material model [122] ......................... 71
Figure 42: Comparison between predicted and experimental behavior of RVE 1 in a)
longitudinal tension and b) longitudinal compression ........................................................................ 73
Figure 43: Comparison between predicted and experimental behavior (modulus) of RVE 2 in a) tension and b) in-plane shear ................................................................. 74
Figure 44: Stress–Strain curves of RVE1. a) Longitudinal tension and compression, b) transverse tension and compression, c) normal tension and compression and d) in- and out-of-plane shear ................................................................. 75
Figure 45: Stress–Strain curves of RVE2. a) Longitudinal tension and compression, b) transverse tension and compression, c) normal tension and compression and d) in- and out-of-plane shear ................................................................. 77
Figure 46: RVE1 longitudinal tension and compression failure modes at the final point of the respective stress-strain curve for the 0 and the 90 plies. ....................................................... 78
Figure 47: RVE1 transverse tension and compression failure modes at the final point of the respective stress-strain curve for the 0 and the 90 plies ....................................................... 79
Figure 48: RVE1 normal tension and compression failure modes at the final point of the respective stress-strain curve for the 0 and the 90 plies ....................................................... 80
Figure 49: RVE1 in- and out-of-plane shear failure modes at the final point of the respective stress-strain curve for the 0 and the 90 plies ....................................................... 81
Figure 50: RVE2 longitudinal tension and compression failure modes at the final point of the respective stress-strain curve for the 45 ply ....................................................... 82
Figure 51: RVE2 transverse tension and compression failure modes at the final point of the respective stress-strain curve for the 45 ply ....................................................... 82
Figure 52: RVE2 normal tension and compression failure modes at the final point of the respective stress-strain curve for the 45 ply ....................................................... 83
Figure 53: RVE2 in- and out-of-plane shear failure modes at the final point of the respective stress-strain curve for the 45 ply ....................................................... 84
Figure 54: Definition of RVE’s geometry [100] ........................................................................ 85
Figure 55: Flowchart of the geometrical modelling of the 3D FIW material ........................................ 85
Figure 56: Representation of a Bezier curve ........................................................................... 86
Figure 57: Definition of the node positions and Bezier splines that describe the tow’s paths..... 87
Figure 58: Definition of slave tows ....................................................................................... 87
Figure 59: Geometrical model of the 3D FIW composite ................................................................. 89
Figure 60: Comparison between cross section of the actual material and geometrical model in
   a) in X-Z plane, b) in Y-Z plane. The images of the 3D FIW’s cross section are
   obtained by CT scans [99] ............................................................................................................... 89
Figure 61: Voxel FE model of the 3D FIW composite material .......................................................... 91
Figure 62: Element coordinate system in tension and compression ................................................. 92
Figure 63: Stress–strain curves for different values of damage parameter m ................................. 94
Figure 64: Comparison between predicted and experimental stress–strain curves for
   different mesh densities. Damage parameter m=2 in all analyses. .............................................. 96
Figure 65: Comparison between predicted and experimental stress–strain curves for
   different values of damage parameter m. Number of elements equals to 128,000 in
   all analyses. ................................................................................................................................. 96
Figure 66: Predicted Stress-Strain Curves of the 3D FIW material. a) Longitudinal tension and
   compression, b) transverse tension and compression, c) normal tension and
   compression and d) in- and out-of-plane shear ........................................................................... 98
Figure 67: 3D FIW longitudinal tension and compression failure modes at the final point of
   the respective stress-strain curve for the matrix, warp and weft tows. .............................. 100
Figure 68: 3D FIW transverse tension and compression failure modes at the final point of the
   respective stress-strain curve for the matrix, warp and weft tows ........................................... 101
Figure 69: 3D FIW normal tension and compression failure modes at the final point of the
   respective stress-strain curve for the matrix, warp and weft tows .......................................... 102
Figure 70: 3D FIW in- and out-of-plane shear failure modes at the final point of the respective
   stress-strain curve for the matrix, warp and weft tows .............................................................. 103
Figure 71: Comparison between experimental and numerical results for the H joint fabricated
   of NCF material .......................................................................................................................... 105
Figure 72: Failure initiation and propagation of the initial geometry of the NCF fabricated H-
   joint ........................................................................................................................................... 107
Figure 73: DVs of the H joining element .......................................................................................... 109
List of Figures

Figure 74: Maximum SI for a) composite material elements and b) adhesive elements for an indicative number of designs .................................................................................................................. 110

Figure 75: Stress plot of the adhesive elements for a) initial geometry and b) optimal geometry ........................................................................................................................................ 111

Figure 76: Comparison of the force-displacement curves between the initial and optimum geometries for the NCF H-shaped bonded joint .............................................................................. 112

Figure 77: Comparison of predicted debonding progression between the initial and optimum geometries of the NCF H-shaped bonded joint .................................................................................. 113

Figure 78: Load-displacement curve of the 3D FIW H-shaped joining element ........................................ 114

Figure 79: Failure initiation and propagation of the initial geometry of the 3D FIW fabricated H-joint ........................................................................................................................................... 115

Figure 80: Maximum SI for a) composite material elements and b) adhesive elements for an indicative number of designs .................................................................................................................. 116

Figure 81: Stress plot of the a) initial geometry’s shear $\tau_{xz}$ stresses and b) optimal geometry’s shear $\tau_{xz}$ stresses ........................................................................................................................................ 117

Figure 82: Comparison of the force-displacement curves between the initial and optimum geometries for the 3D FIW H-shaped joining element .............................................................................. 118

Figure 83: Comparison of predicted debonding progression between the initial and optimum geometries of the 3D FIW H-shaped joining element .......................................................................... 120
Table 1: Material properties of EA9695 film adhesive [54] ................................................................. 44
Table 2: Sudden property degradation concept ......................................................................................... 49
Table 3: Defined properties for various applications in WiseTex package .................................................. 59
Table 4: Material and mechanical properties of the fiber Toho Tenax F-13 ................................................. 63
Table 5: Layer properties of the NCF RVEs .................................................................................................. 63
Table 6: Toho Tenal HTS/ RTM6 (V_f=55%) material properties ................................................................. 66
Table 7: RTM 6 material properties ............................................................................................................. 67
Table 8: Degradation Rules .......................................................................................................................... 70
Table 9: Calculated properties for RVE1 .................................................................................................... 76
Table 10: Calculated properties for RVE2 .................................................................................................. 77
Table 11: Color pattern of failed elements .................................................................................................. 78
Table 12: Values of volume fractions for the 3D FIW RVE ........................................................................ 88
Table 13: Toray T700/ Reichhold Dion 9500 (V_f=72%) material properties ............................................... 90
Table 14: Reichhold Dion 9500 material properties .................................................................................... 91
Table 15: Comparison between predicted and numerical properties of the 3D FIW material .... 97
Table 16: Comparison between predicted mechanical properties of the 3D FIW composite material and the properties of the UD AS4/3501-6 ply [129] ................................................................. 99
Table 17: Color pattern of failed elements .................................................................................................. 99
Table 18: Color pattern of failed elements ................................................................................................. 108
Table 19: Initial and final optimization parameters for the NCF fabricated H-joint ......................... 111
Table 20: Initial and final optimization parameters for the 3D FIW fabricated H-joint ..................... 117
# List of Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D:</td>
<td>Two Dimensional</td>
</tr>
<tr>
<td>3D:</td>
<td>Three Dimensional</td>
</tr>
<tr>
<td>ABC:</td>
<td>Artificial Bee Colony</td>
</tr>
<tr>
<td>CDM:</td>
<td>Continuum Damage Model</td>
</tr>
<tr>
<td>CFRP:</td>
<td>Carbon Fibre Reinforced Plastic</td>
</tr>
<tr>
<td>CT:</td>
<td>Computer Tomography</td>
</tr>
<tr>
<td>CZM:</td>
<td>Cohesive Zone Modelling</td>
</tr>
<tr>
<td>DCB:</td>
<td>Double Cantilever Beam</td>
</tr>
<tr>
<td>DV:</td>
<td>Design Variable</td>
</tr>
<tr>
<td>FE:</td>
<td>Finite Element</td>
</tr>
<tr>
<td>FEA:</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>FEM:</td>
<td>Finite Element Modelling</td>
</tr>
<tr>
<td>FIW:</td>
<td>Fully Interlaced Woven</td>
</tr>
<tr>
<td>FRP:</td>
<td>Fibre Reinforced Plastic</td>
</tr>
<tr>
<td>GA:</td>
<td>Genetic Algorithm</td>
</tr>
<tr>
<td>HTS:</td>
<td>High Tensile Strength</td>
</tr>
<tr>
<td>NCF:</td>
<td>Non-crimp Fabric</td>
</tr>
<tr>
<td>OF:</td>
<td>Objective Function</td>
</tr>
<tr>
<td>PDM:</td>
<td>Progressive Damage Modelling</td>
</tr>
<tr>
<td>PSO:</td>
<td>Particle Swarm Optimization</td>
</tr>
<tr>
<td>RVE:</td>
<td>Representative Volume Element</td>
</tr>
<tr>
<td>SI:</td>
<td>Stress Index</td>
</tr>
<tr>
<td>SV:</td>
<td>State Variable</td>
</tr>
<tr>
<td>SO:</td>
<td>Shape Optimization</td>
</tr>
<tr>
<td>UC:</td>
<td>Unit Cell</td>
</tr>
<tr>
<td>UD:</td>
<td>Unidirectional</td>
</tr>
</tbody>
</table>
In the present thesis a novel numerical approach for the optimization of composite structures fabricated from woven composite materials is developed. The aim is to increase the ultimate strength of the structure while at the same time decreasing its weight. The numerical approach is based on a combination of the numerical algorithm of PDM, along with SO in an iterative subroutine. PDM, which is comprised of three steps, namely stress analysis, failure analysis and material property degradation, is used to predict the initiation and propagation of failure in the structure. During the phase of SO certain geometrical parameters are varied within limits in order to minimize the stresses that lead the structure to ultimate failure as indicated by PDM results. Finally the resulting geometry is solved with PDM to ensure the enhancement in the ultimate strength and the decrease in ultimate weight.

Within the frame of this approach, a new methodology for the numerical modeling and the simulation of mechanical behavior of woven composite materials is proposed. The highly inhomogeneous nature of woven composite materials in the micro-scale is taken under consideration to create accurate RVE FE models which represent the actual material. Then PDM is used for the simulation of their mechanical response. The calculated properties, in terms of stiffnesses and strengths, are then inserted as inputs in the global FE model of the composite structure. Additionally, the reliability and applicability of a CDM, in comparison with CZM, are assessed in order to use the CDM for the modeling of the adhesive’s mechanical behavior.

The mentioned numerical approach is applied in an H-shaped joining element fabricated from two different woven composite materials for the loading case of tension. In the first case NCF composite is used while in the second case the joint is made of 3D FIW composite. The purpose of the H-shaped element is the joining of two composite plates via the method of adhesive bonding.
Η ανάπτυξη νέων υλικών που θα οδηγήσουν σε ελαφρύτερες και λιγότερο επιρρεπείς στην βλάβη δομές, είναι σταθερό ζητούμενο για την αεροπορική βιομηχανία. Την τελευταία δεκαετία δεκαετία με αεροσκάφη όπως το Airbus A380 και κυρίως το Boeing 787 η μετάβαση από τις μεταλλικές δομές σε δομές συνθέτων υλικών έχει εν μέρει επιτευχθεί με ποσοστά σε σύνθετα υλικά μεγαλύτερα από 50%. Αυτό μειώνει σημαντικά το βάρος του αεροσκάφους με αποτέλεσμα την μειωμένη κατανάλωση καυσίμου. Μια πολλά υποσχόμενη κατηγορία συνθέτων υλικών είναι τα πλεκτά σύνθετα υλικά, στην οποία ανήκουν τα μη-πτυχωτά σύνθετα (Non-Crimp Fabrics) και τα τρισδιάστατα πλεγμένα υλικά (3D Woven) που αναμένεται να μειώσουν το πρόβλημα της διαστρωματικής αποκόλλησης λόγω των ενισχυμένων εκτός-επιπέδου ιδιοτήτων τους. Εντούτοις, οι τρέχουσες μέθοδοι σύνδεσης δομικών στοιχείων παρουσιάζουν σημαντικά μειονεκτήματα που σε πολλές περιπτώσεις αναφέρουν το όφελος από την χρήση προηγμένων υλικών. Ως γίνεται εύκολα κατανοητό, η βελτιστοποίηση των δομικών στοιχείων από σύνθετα υλικά κρίνεται απαραίτητη για την χρήση τόνωσε και την κατανόηση της μηχανικής συμπεριφοράς των εξαιρετικών ειδικών ιδιοτήτων που αυτά προσφέρουν.

Στην παρούσα διατριβή αναπτύχθηκε μια νέα μέθοδος αριθμητικής βελτιστοποίησης δομικών στοιχείων από σύνθετα υλικά με σκοπό την αυξημένη της αντοχής τους. Η μέθοδος βασίζεται σε έναν αριθμητικό άλγοριθμο Προοδευτικής Εξέλιξης της Βλάβης (ΠΕΒ) και τη Βελτιστοποίηση Σχήματος (ΒΣ) τα οποία συνδυάζονται σε μια επαναληπτική υπο-ρουτίνα. Στην ΠΕΒ περιλαμβάνονται τα βήματα της ανάλυσης τάσεων, ανάλυσης αστοχίας και υποβάθμιση των ιδιοτήτων των στοιχείων. Η χρησιμότητα της έγκειται στην πρόβλεψη της έναρξης και εξέλιξης της αστοχίας στο δομικό στοιχείο κάτι απαραίτητο για την κατανόηση της μηχανικής συμπεριφοράς. Η ΒΣ εγκειόνται στον μεταβολή συγκεκριμένων γεωμετρικών παραμέτρων για να επιτευχθεί ελαχιστοποίηση των κρίσιμων τάσεων που προκύπτουν από αποτελέσματα της ΠΕΒ και οδηγούν στην αστοχία του στοιχείου. Παράλληλα, για την επέκταση και τον υπολογισμό των
μηχανικών ιδιοτήτων πρωτότυπων πλεγμένων σύνθετων υλικών προτείνεται καινούργια μια μεθοδολογία η οποία λαμβάνει υπ’ όψιν την υψηλή ανομοιογένεια των υλικών στην μικρό-κλιμάκα για να υπολογίσει τις ιδιότητες τους. Για την προσομοίωση της μηχανικής τους συμπεριφοράς χρησιμοποιήθηκε η μεθοδολογία της ΠΕΒ.

Η μεθοδολογία εφαρμόστηκε σε ένα νέο συνδετικό στοιχείο σχήματος Η κατασκευασμένο από δύο διαφορετικά πλεγμένα σύνθετα υλικά, τα μη πτυχωτά και τα τρισδιάστατα πλεγμένα σύνθετα υλικά, για την περίπτωση του εφελκυσμού. Σκοπός του συνδέσμου είναι η ένωση δύο πλακών από σύνθετα υλικά χρησιμοποιώντας κόλλα.

Αρχικά το μοντέλο πεπερασμένων στοιχείων του συνδέσμου δημιουργείται και επιλύεται με την μέθοδο ΠΕΒ. Η προσομοίωση της βλάβης στα σύνθετα υλικά γίνεται με την επιλογή κατάλληλων κριτηρίων αστοχίας και κανόνων υποβάθμισης των ιδιοτήτων των υλικών. Για την προσομοίωση της μη-γραμμικής συμπεριφοράς της κόλλας αναπτύσσεται ένα δι-γραμμικό μοντέλο. Η δυνατότητα του μοντέλου αυτού επικυρώνονται μέσω της χρήσης του για την πρόβλεψη της συμπεριφοράς δοκιμών διπλής αμφιέρειστης δοκού. Για την προσομοίωση της πλήρης μηχανικής συμπεριφοράς των μη πτυχωτών και τρισδιάστατα πλεγμένων συνθέτων υλικών, αναπτύσσεται μια διαδικασία η οποία περιλαμβάνει τα βήματα της γεωμετρικής μοντελοποίησης, της κατασκευής του μοντέλου πεπερασμένων στοιχείων και την επίλυση αυτού με την μέθοδο ΠΕΒ. Η γεωμετρική μοντελοποίηση διεξάγεται με κατάλληλα προγράμματα επεξεργασίας γεωμετρίας και ακολούθως, το γεωμετρικό μοντέλο μεταφράζεται σε κώδικα πεπερασμένων στοιχείων για επίλυση. Η συμπεριφορά των στρώσεων προσομοιώνεται με κατάλληλα κριτήρια αστοχίας και εξισώσεις βλάβης ενώ η μη-γραμμική συμπεριφορά της στρώσεως πεπερασμένων στοιχείων για επίλυση αυτού με την μέθοδο ΠΕΒ. Η συμπεριφορά των στρώσεων προσομοιώνεται με κατάλληλα κριτήρια αστοχίας και εξισώσεις βλάβης ενώ η μη-γραμμική συμπεριφορά της στρώσεως πεπερασμένων στοιχείων για επίλυση αυτού με την μέθοδο ΠΕΒ.
Στην συνέχεια, λαμβάνει μέρος η γεωμετρική βελτιστοποίηση βασιζόμενη στα αποτελέσματα της επίλυσης της αρχικής γεωμετρίας. Σε αυτό το σημείο επιλέγεται η μεταβλητή προς ελαχιστοποίηση στην διαδικασία της βελτιστοποίησης. Το μέγεθος αυτό ονομάζεται Συνάρτηση Σκοπού (ΣΣ) και ορίζεται ως ο συντελεστής βλάβης που ευθύνεται για την τελική αστοχία του δομικού στοιχείου. Ως ένα επιπλέον κριτήριο για την επιλογή της βέλτιστης γεωμετρίας επιλέγεται η μείωση βάρους δεδομένου ότι πρόκειται για αεροπορική κατασκευή. Για την επίτευξη των προαναφερθέντων στόχων οι γεωμετρικές παράμετροι που υπόκεινται σε μεταβολή είναι μόνο εκείνες που δεν επηρεάζουν τις εξωτερικές διαστάσεις του δομικού στοιχείου. Η γεωμετρία που ελαχιστοποιεί την συνάρτηση σκοπού και ταυτόχρονα είναι αρκετά ελαφρότερη από την αρχική, επιλέγεται ως η τελική γεωμετρία. Τέλος, γίνεται η επικύρωση της βελτιστοποίησης με την σύγκριση των αριθμητικών αποτελεσμάτων μεταξύ της αρχικής και τελικής γεωμετρίας. Η μεθοδολογία της ΠΕΒ εφαρμόζεται στην τελική γεωμετρία και τα διαγράμματα δύναμης μετατόπισης συγκρίνονται για να διαπιστωθεί η αύξηση στο μέγιστο φορτίο που μπορεί να φέρει το συνδετικό στοιχείο πριν την τελική αστοχία.
1.1. **The Technological Problem**

Prior to the mid-1980s, the aerospace industry was using composite materials in secondary structures such as wing edges and control surfaces of commercial aircraft. The reasons why composite materials in commercial air transport have always lagged behind the expectations of the aircraft industry but also behind military usage lie mainly to the insufficient knowledge about the mechanical and long term of these materials which results to the need for more conservative considerations as well as to expensive manufacturing techniques which are necessary in order to ensure the quality required by the aircraft industry. The first aircraft to include composite structures in primary applications were the Airbus A320 in 1988 and the Boeing 777 in 1995. They both featured a composite tail section. Since then, a growth in percentage of the use of composites has been realized, resulting in aircraft like the Airbus A 380 and A350 (22%, 52% composite materials respectively) and the Boeing 787 (50% composite materials). This ongoing trend of increasing the use of composites in airplanes is captured in Figure 1. As a consequence, research must now focus in new areas that are necessary for, accomplishing a safe transition from metallic to composite aircraft parts. One such focal area is the efficient joining of composite structures.

To this date the established method for joining composite structures is mechanical fastening. Nevertheless, an undesirable weight penalty is introduced owing to the accessional weight of the bolts. What is more, due to the machining required to drill the holes in the composite parts initial damage may take place which, in conjunction with the low out of plane properties of the typical UD composites, can lead to delamination and ultimately, to fatigue-induced premature failure. To solve this problem extra material around the hole is added, but this further increases the total weight. Alternatively, adhesive bonding which was developed as
a direct alternative to riveted joining, has been used for more than 40 years in aircraft structures [1] to join metallic parts. Nonetheless, to establish adhesive bonding as a reliable fastening method, its ability to transfer the required load between the assembled parts must be ensured. The low normal strength of most adhesives has led to parts re-design through involving shapes, such as H, Π, L, where the load is transferred mainly through shear stresses. On the other hand, with the specific design of bonded joints high normal stresses arise in specific areas. It should be noticed that traditional UD composites, widely used in aerospace applications, suffer from matrix-dominated failures (e.g. delamination and matrix cracking) due to the relatively low fracture toughness of the matrix. These failures make UD composites prone to fatigue and impact [2, 3] when high out-of-plane stresses arise. In the last decade, an unremitting research effort is in progress to increase the matrix-dominated mechanical properties of the composite

Figure 1: Commercial aircraft models and the percentage of used composite materials
materials [4]. In this frame, new material architectures such as NCF and 3D FIW composite materials with enhanced out of plane properties have been proposed, offering a promising alternative to the currently used composite materials.

While optimization of mechanically fastened joints has reached a limit, its drawbacks are still present. Consequently, the process of introducing new concepts to the area of joining composite structures as well as new composite material configurations has been initiated [5]. To this end, optimization of adhesively bonded joints is a basic function of the redesign process required for the realization of adhesive joining as a consistent assembly method. Moreover, the exploration of the mechanical response and failure mechanisms of novel material architectures will facilitate the substitution of traditional composite materials. Yet, as this effort is at an early stage, many obstacles need to be overcome; a costly and time consuming task. As a direct result, apart from the experimental work, dependable FE models must be developed in order to reduce the study time and cost. To assist the re-design process of composite joining elements, the proposal of integrated numerical optimization approaches is imperative. Additionally, developing comprehensive methods for predicting the mechanical response of advanced composite materials will be a decisive step toward their introduction as trustworthy choices in aerospace applications.

To this date many European research programmes have attempted to address the aforementioned challenges with the active collaboration between aeronautical industries, technological institutes, and universities. An important part of the research included in this work has been conducted within the framework of the European research program CERFAC [5]. This program focuses on the development and adoption of adhesive bonding between composite parts and new composite material architectures. These two research fields are expected to give solutions to some of the most fundamental issues discussed above and lead to cutting-edge technologies.
1.2. **AIM AND APPROACH**

The aim of this thesis is to introduce a numerical approach for the shape optimization of composite structures maximizing their ultimate strength while reducing their total weight. The work focuses on composite structures fabricated with novel woven fabric composite materials.

The numerical approach utilizes PDM and SO, which are combined in an automatic process to propose lighter and stronger composite structures. The use of PDM lies in predicting the initiation and propagation of damage to gain better understanding about the mechanical response of the structure. The target of SO is to propose an optimal design by altering specific geometric parameters within limitations such as weight, based on the PDM results. Alongside, for the prediction of the mechanical response of woven composite materials a modelling process is proposed which accounts for the inhomogeneity in their micro-scale to calculate their macroscopic mechanical properties.

To achieve the aim of the thesis the following steps have been followed:

i) Development of an easily applicable methodology for the FE modelling of woven composite material while considering their highly inhomogeneous nature
   
   • The methodology takes into account the specific geometrical characteristics of woven materials and lead to an accurate FE model
   
   • This methodology has been applied to NCF and 3D FIW to calculate their mechanical properties in terms of stress-strain curves for all normal and shear loading cases
   
   • The methodology has been successfully validated via comparison between experimental and numerical results available in the literature

ii) Development of a 3D FE model of the composite structure and assignment of the calculated composite material properties to its elements

   • The PDM concept was applied to simulate the mechanical response of the structure under specific load-cases
• Validation of the FE model of the structure was made by comparing numerical with experimental results available in the literature

iii) SO was performed in order to increase the load carrying capacity of the structure and reduce its weight

• Verification of the optimization has been carried out by comparing numerical results between initial and optimized geometries

iv) Validation of the numerical approach was performed applying it on an H-shaped multifunctional joining element fabricated of woven fabric composite materials

1.3. **Thesis Summary**

Chapter 1: **Subject of the Thesis**

The initial chapter is divided in three parts. Initially the technological problem of the present thesis is discussed. Subsequently the aim of the thesis and the approach to achieve it is presented. Finally in this sub-chapter, a brief summary of each chapter is given.

Chapter 2: **Literature Review**

In the first part, the optimization of composite structures is covered. Initially, the most common optimization algorithms are presented. Then the progress in the fields of FE modelling in conjunction with optimization methods and optimization of bonded joints are reviewed. The second part is dedicated to woven composite materials. Following a review of different woven material architectures, the state-of-the-art in FE modelling is presented.
Chapter 3: Definition of Basic Terms

In this chapter basic terms of this thesis are explained to help the reader gain a better understanding of the information included in this thesis.

Chapter 4: Development of the optimization methodology for composite structures

The third chapter is divided into three parts where the different sections of the optimization methodology are introduced. Initially, the concept of PDM, which contain the steps of stress analysis, failure analysis and material property degradation, is analysed. Afterwards, the optimization process is described. The optimization variables and the optimal design selection technique are presented. The step of validating the proposed optimization methodology is then briefly discussed.

Chapter 5: Application of the methodology to the H-shaped joining element

The proposed methodology is applied to an H-shaped joining element fabricated of woven composite materials. At the outset, the PDM concept is explained with respect to the component of the H-shaped joining element. Subsequently, a methodology for the modelling of woven composite materials is developed. Then, the concept is applied on NCF and FIW materials in order to predict their mechanical response.

Chapter 6: Numerical characterization of woven composite materials

In this chapter the results of the mechanical characterization of the materials are presented. The chapter contains two main sections, one for the NCF and one for the FIW material. In both cases, the results are divided into three parts. In the first part, the validation of the FE model is presented by comparing experimental
and numerical results. Then the damage initiation and propagation is captured. The final section contains the stress-strain curves for each material.

Chapter 7: **Optimization results**

In chapter six the optimization results are presented for both material configurations. Firstly, the FE model of the joining element is validated through comparison with experimental results. The following sections contain the SO results, as well as the final comparison between the initial and the final geometries which validates the proposed optimization methodology.

Chapter 8: **Conclusions**

Summary of the thesis along with the main conclusions resulting from this work. The achieved results are discussed in relation to the initial aim and objectives.

Chapter 9: **Future Work**

In this chapter some recommendations for future research are made.
2.1. **Optimization of Composite Structures**

Engineers seek the best possible design in every construction be it a civil engineering structure or a space shuttle. Structures based on composite materials constitute no exception. Optimization is a field of mathematics often involved in decision guidance [6]. In this field, one wishes to minimize or maximize a function in order to guide the decision-maker to the best choice. According to the specific needs of an application, a possible design is usually evaluated in terms of mechanical properties (stiffness or strength) or available resources (weight or cost) [7]. Consequently some applications require minimization of weight or cost while maintaining a specific strength or stiffness whilst others predefine a cost or weight limit to achieve the best mechanical performance.

Optimization oftentimes, is a lengthy process to reach to the desired outcome. If more than one objectives are to be met, many iterations are often needed, especially for cases where the objectives are conflicting. Thereby, it can be easily understood that optimization performed by experiments is a tedious and costly process especially if done by ‘trial and error’. Fabricating and testing structures with different material and geometrical configurations is time consuming and requires a great deal of resources violating cost limits. On the contrary, numerical optimization has emerged as a powerful tool for structural design. Within the framework of the optimization procedure, the desirable quantity to be minimized or maximized is set as the OF. This can be set according to the needs of a specific application either in terms of mechanical performance or in terms of weight or cost. Then, independent variables such as geometrical or material parameters are varied to achieve the best possible result. Compared to experimental procedures, the clear advantage of numerical optimization is the significant reduction on the usage of resources it offers.
Up to date there many special optimization algorithms have been developed which deal with single or multi optimization problems. Among the most used are the GA [8, 9, 10, 11], PSO [12, 13] and ABC [14, 15] algorithms. However, these algorithms alone cannot handle the optimization of complex composite structures. It is indicative that most works that use such algorithms without any kind of post processing to evaluate the optimal solution, focus on simple composite plates to minimize cost or internal stresses. Since it is beyond the scope of this work to examine in depth the pros and cons of each algorithm, we will focus only on works that discuss the optimization of complex composite and ideally combine optimization procedures with FEM.

As early as 1980s, NASA developed [16, 17], a computer code, named PASCO (Panel Analysis and Sizing Code) which could be used for buckling analysis and sizing of composite panels. The code provided a base upon which, Nagendra et.al [18] combined PASCO with FEM using a sequential approximate optimization approach to design a blade stiffened panel with a hole. The optimization procedure is based on simultaneous use of the PASCO panel buckling analysis and sizing code together with the Engineering Analysis Language FEA code EAL. Material and geometrical parameters can be chosen as DVs to achieve an optimal design in terms of mechanical response. After a decade, a multi-objective optimisation computer program, MOST (Multifactor Optimisation of Structures Technique), was developed to treat complex engineering problems which can have multiple objectives and DVs [19]. The MOST system uses FE codes (e.g. ABAQUS or ANSYS) as its analysis program, and its optimization system control program is written in UNIX shell scripts. The program developed was used to optimize antenna performance in terms of surface accuracy, structural strength, stiffness, natural frequencies, and structural mass, while considering the thermal deformations due to the temperature changes in space environment and in a launch case. Even though it cannot be assured that the method is capable of finding the global optimum, it showed the potential offered by the combination of FEA and optimization methods to handle difficult engineering optimization problems. The initial and optimal design of the antenna is depicted in Figure 2. The MOST system is also used in [20] to determine the optimal topology of the truss-core panel under different loading conditions where the same conclusions as in [19] were drawn.
A different, two-level approach, for the simplification of intricate engineering structures is presented in [21] and further discussed in [22]. In [21] the authors optimize a wing structure subjected to strength and buckling constraints. The problem is decomposed in two levels, the panel and the wing level, each one with its distinctive DVs and OFs. The procedure is based on continuous optimization at the wing level using an FE model, and genetic optimization at the panel level. Communication between the two levels is based on a response surface of optimal panel buckling load. The wing-level optimization is carried out by the GENESIS program using the response surface optimization results. As a final point, the procedure proved to be successful. This procedure is generalized in [22], and is applied to global-local structural optimization problems. The global-local optimization approach can be found in a different form in [23]. This work uses the Bi-directional Evolutionary Structural Optimization (BESO) method, a two-scale topology optimization algorithm, for the parallel optimization of macrostructures and their respective composite microstructures. FEA is used to obtain sensitivity numbers at the macro- and micro-scale levels and then BESO method is applied to update the macrostructures and the composite microstructures according to the elemental sensitivity numbers at both scales. The authors conclude that this method can provide design than is superior to that based on a one-scale optimization, as the design degrees of freedom are increased significantly. The coupling between GA algorithms and FEA, as a tool for the calculation of the OFs, in an iterative process is developed in [24]. The authors optimize a composite plate in the sense of
stacking sequence, shape and size. To calculate the stresses in the composite plate and evaluate the candidate designs, they use FEA which, in conjunction with the optimization algorithm, is capable of successfully concluding to an optimal solution. However the FE package in this work is treated as a “black box” where no modifications can take place to account for different type of problems. The flowchart of the methodology proposed in [24] is presented in Figure 3. Also in [25], in which the mechanical performance and manufacturing cost is optimized, FEA is employed to predict the displacement under given loading conditions. Besides that, for the mould filling analysis, FEA along with a semi-analytic model is utilized. It is nevertheless concluded that while FEA can provide accurate results for mould filling time evaluation and handle complex mould geometries, its computational cost can sometimes be prohibitive for optimization purposes.

![Flowchart of the methodology proposed in [24]](image)

Figure 3: Flowchart of the methodology that combines FEA and special optimization algorithms proposed in [24]

From the overview of the literature about optimization of composite structures the following conclusions can be drawn:

- Lack of numerical approaches for the optimization of complex structures that can be applicable to any composite structural part
- Optimization procedures are used almost exclusively to simple complete structures such as composite plates
- Strength is rarely taken into account. Instead most works focus on the reduction of cost while keeping a minimum ultimate strength
- Optimization in conjunction with PDM as a mean for evaluating the optimal solution is non-existent

2.2. **Optimization of Bonded Joints**

Adhesive bonding was developed as an alternative method to mechanical fastening mainly for joint metallic components. However, as the usage of composite structures has been increased in recent years, it was inevitable that the implementation of adhesive bonding to composites would take place. The structural integrity of adhesively bonded composite parts depends on the magnitude of stresses developed in the adhesive and the adherents. Therefore, geometry optimization of the bonded joints is required to keep these stresses as low as possible, and establish adhesive bonding as a reliable joining method.

Some of earliest works that studied the stress distribution in adhesive joints [26, 27, 28, 29] can be traced back to the 1970s. Ojalvo [28] and Cherry and Harrison [29], discuss how to shape the adherents in order to obtain a constant shear stress in the adhesive layer. Nonetheless, the optimization of bonded joints using the FE method did not start until 1991 with the work of Groth and Nordlund [30]. In this paper the authors implement the structural shape optimization program OASIS-ALADDIN to examine different ways to optimize the geometry of bonded joints and structures fabricated from aluminium. The objective of this study is to acquire lighter and stronger joints. The joint types studied are single and double lap joints and double-strap joint. Even though the optimization proved to be successful, the authors questioned the possibility of fabricating the proposed ‘optimal shapes’. In particular, they concluded that it was very difficult to accomplish the correct profiling due to machining restrictions. The geometrical differences of a single-lap joint can be seen in Figure 4.
Non-linear FE methods are applied to the analysis of single lap joints between FRP and metals in [31]. The optimization is performed with regard to strength, while treating the geometrical characteristics as DVs. More specifically, different shapes of adhesive fillet, reverse tapering of the adherent, rounded edges and denting were tested to increase the joint’s ultimate strength. The conclusion is that both geometric and material nonlinearities affect the stresses developed at the joints. Moreover, the authors outline the need for additional testing regarding the mechanical properties of structural adhesives. In [32], an iterative optimization method using the FE code PAFEC is developed. The study focuses on problems related to life extension of aircraft components. In order to achieve constant boundary stresses, the nodes are moved on the stress concentration boundary. The flowchart of the method is presented in Figure 5. The results indicate that reduction in peak stresses can reach 30%.

The optimization of adhesive fillets is studied in [33]. The Evolutionary Structural Optimisation Method (EVOLVE) is employed to optimise the shape of adhesive fillets found in tabs of tensile test specimens. This method relies on an iterative FEA and progressive removal
of elements to minimise the maximum stresses in the joints. The achieved reduction in maximum principal stress ranged between 48 to 64%.

![Flowchart of the methodology used in [32]](image)

**Figure 5: Flowchart of the methodology used in [32]**

Typical optimization analyses of bonded joints consider only the shape of adherents as DV to achieve smooth transition of stresses. On the other hand, in the works of Kay and Heller [34, 35] the thickness of the adhesive is perceived additionally as a parameter subjected to change aiming at reducing stresses in the adhesive itself. In [35], an automated sensitivity-based
shape optimisation procedure is developed for the optimal design of free-form bonded repairs and lap-joints, to achieve reduced adhesive stresses. The approach is applied to a number of single and double-sided configurations where both the shapes of the adhesive layer and the outer adherent were conceived as DVs. The authors comment that reduction in adhesive stresses after the optimization is sufficient to keep them in the elastic range, severely reducing the possibility of fatigue failure. The same authors in [36] performed through-thickness optimization of double-lap joints considering also the effects of differential thermal contraction during curing. The configuration of the joint studied in [35] [36] is depicted in Figure 6.

![Figure 6: Double lap joint configuration studied in [35] [36]](image)

The use of genetic algorithms in conjunction with single and multi-objective functions for the optimisation of a structural steel/composite connection is proposed in [37]. Although the results from the single objective method prove very successful in reducing weight, stress and stiffness, the multi-objective method provides an even better joint configuration reducing both weight and stress in the adhesive by a substantial amount compared to the baseline joint. Contrary to the most optimization works which perform simple stress analyses, [38] implements a CDM concept to simulate the failure path of the adhesive. This work presents a two-dimensional numerical analysis to assess the influence of several geometric changes on the tensile residual strength of repaired CFRP composite plates. The geometric changes included chamfering the patch outer face, thickening the adhesive near the overlap outer edge, filling the plate’s gap with adhesive (plug filling), using fillets of different shapes and dimensions at the
patch ends, chamfering the outer and inner plate edges, and combinations of these. The results prove that with correct joint configuration, the residual strength can be increased by 27% for single-lap joints and 12% for double-lap joints. Also good agreement is found between experimental and numerical predictions of failure path and load. Indicatively, the predicted failure path of the adhesive for the case of filleted joint is presented in Figure 7. The CZM approach is used in [39] to predict the mechanical response of spot-welded/bonded single-lap joints under geometrical changes. A parametric study on the overlap length (LO) allowed achieving significant strength advantages, up to 58% compared to spot-welded joints and 24% over bonded joints.

![Figure 7: Numerical prediction of the failure path of the adhesive for a filleted joint [38]. The crack starts at the tip of the filleted joint](image)

From the overview of the literature about optimization of bonded joints the following conclusions can be drawn:

- Most studies focus on joints with metallic adherents
- Research mainly on simple single and double lap joints
• Weight or dimensional limits are seldom taken into account. As a result, the optimized shape is often difficult to produce
• Evaluation of damage in the adherents in not considered

2.3. **JOINING ELEMENTS**

Initially composite joining elements were developed in the early 1990s for maritime applications. Joining elements come in various shapes depending on the application. Characteristic shapes of composite joining elements as well as indicative applications may be seen in Figure 8a, b. Their primary function is to transmit tensile shear and flexural loads between the adherents

![Figure 8: a) Illustration of differently shaped joining elements and b) application of Pi-shaped joining element](image)
The first work to assess the ability of a T joint to transfer out-of-plane loads is presented [40]. In this work experimental results are compared with analytical solutions which can predict the failure load. Based on experimental conclusions a suitable design for specific maritime applications is developed. A study on how different material and geometrical characteristics affect the performance of T-joints studied in [40] is conducted in [41]. Apart from the mechanical testing, FE models are generated to provide information about the internal load dissipation and failure mechanisms. The modelling is used to investigate influences of geometry and material variations on the performance of the tee joints. Various parameters were taken into consideration, which are presented in Figure 9. The geometrical variables that affect the most the mechanical response of the joint are found to be the radius of the fillet and the thickness of the over-laminate. The T-joint studied in [41] is presented in

Figure 9: Illustration of a) the parts of a T-joint and b) optimization parameters studied in [41]

In [42] a mechanical testing on T-joint used in high speed marine vehicles is conducted. Also, non-linear FE models are developed to study the areas of high stress concentration and the influence of geometric parameters. A review performed in [43] about the major factors in the performance of T-joints indicates the need for further work in the areas of accurate
measurement of basic material data, theoretical methods used for design, static testing and characterization, and fatigue and impact performance. The mechanical response of transverse stitched T-joints in bending is investigated in [44]. FEA is performed on the structural joint to calculate failure loads. Experiments are conducted to determine the modes of failure and ultimate failure strength. Moreover, parameters such as local web thickness, flange thickness, number of rows of stitching, and resin types were varied to determine the joint performance. To enhance the performance of the joints the authors suggest specific modifications. The same authors perform parametric analyses of stitched composite T-joints [45]. In this work, the effect of parameters such as fiber insertion tow modulus, fiber insertion filament count, fiber insertion depth, and resin-rich interface zone thickness on T-joint displacement and damage initiation load is evaluated. The results indicate that, under flexure loading, increasing the fiber insertion tow modulus and filament count increases the initial damage load. Also, increasing the fiber insertion depth reduces T-joint deflection and improves the damage initiation load and reducing the interface thickness reduces the T-joint deflection and improves the damage initiation load. Some more recent works on T joints include [46, 47, 48, 49]

Another similar type of joining element is the Pi-joint. Among the first studies to examine its mechanical performance are [50, 51]. In [50] the behaviour of a composite Pi-joint under a static, tensile load is investigated. The experimental setup is presented in Figure 10. Furthermore an FE model capable of predicting its damage onset, propagation, and ultimate collapse is created. The FE model implements the PDM and the numerical and experimental results are compared with good agreement between them.

A mesomechanical approach for simulating the mechanical performance of NCF composite structural parts is presented in [52]. The authors develop RVEs to calculate the properties of NCF material. Subsequently these properties are used as inputs in a macro scale model of a Pi-joint to simulate its mechanical response. The results indicate that tensile extensive failure at the bottom of the joint lead to catastrophic failure of the structure. A study on the effects of different failure criteria in the simulation of Pi-joints is performed in [53]. Analyses with four different failure criteria are performed and compared to experimental results. The authors
conclude that the modified maximum stress failure criterion is suitable for the filler zone of the joint, and the Hashin failure criterion for the other zone of the joint.

Figure 10: Pi-joint and experimental setup used in [50]

While T-joints, and to a lesser extent Pi-joints, have been studied systematically, this is not the case for H-joints where the available literature is almost non-existent. To the author’s knowledge only two works [54], [55] have been reported in this area. The process of fabrication an H-joint is explained in [54]. A typical H-joint can be seen in Figure 11. In [55], the mechanical response of H-joints in tension shear and bending are investigated. Additionally, an FE model capable of predicting damage initiation and propagation of failure in the adhesive and the adherents is developed.

Figure 11: Cured H-joint [54]
From the overview of the literature about joining elements the following conclusions can be drawn:

- Studies of the H-shaped joining elements are very few
- No optimization procedures have been applied in joining elements

2.4. **Woven Composite Materials**

A composite material can be regarded as any material which consists of two or more dissimilar components in order to fabricate a final product with properties exceeding those of each component alone. While this technique has been used for many thousands of years, it was not until the early 1960s that fibrous composites as a suitable engineering material, was instigated with the development of carbon fibres in the UK, and boron fibres in the USA [56], in FRP composites. Usually composite materials consist of the matrix and the reinforcement. CFRP are composites with polymeric matrix which use carbon fibres as reinforcement to boost the matrix’s mechanical properties. Traditional UD composites, widely used in aerospace applications, suffer from matrix-dominated failures (e.g. delamination and matrix cracking) due to the relatively low fracture toughness of the matrix. These failures make UD composites prone to fatigue and impact [57, 58] when out of plane or in-plane compresive loadings are dominant. In the last decade, a continuous research effort is ongoing in order to increase the matrix-dominated mechanical properties of composite materials [59]. In this framework, new material configurations have been proposed.

Woven composite materials are a subclass of composites where the reinforcement is a textile material consisting of fibres, typically arranged as tows or yarns. Depending on the existence of tows in the 3rd direction (or Z axis) woven fabrics are divided in 2D and 3D woven. In 2D category, many different types can be found such as plain, twill weaves and Non-Crimp Fabrics. 3D woven fabrics can be found in three main variations namely angle interlocked, orthogonally interlocked, and fully interlocked.
2.4.1. **NCF Composite Materials**

Initially the NCF materials were developed in 1983 [60] for applications in the marine industry in terms of a +45° ply knitted together with a -45° ply to form a double bias fabric. As their name states, NCFs, shown in Figure 12 and Figure 13 differ from woven fabrics in that, UD layers of fibres are held together by through thickness stitching. Usually the stitching thread is made of polyester and two or four layers are stitched together. Consequently there is no interlocking of fibre tows. This is reflected in their mechanical properties which are not significantly below those of UD composites. Yet, the main reason why NCF materials are chosen instead of UD in many industrial applications has to do with cost. According to Bibo et al. [61], 35% reduction of the cost was observed when NCF composites with resin transfer molding were used instead of UDIs fabricated in an autoclave. In particular, the labor required to align and unbraid the fibers in a UD composite can reach up to 50% of the total cost.

![Stitching Pattern](image)

Figure 12: Representation of 0/90 NCF preform where the stitching pattern and the different orientation of the tows can be seen.
The main purpose served by NCF materials is the effective enhancement of out-of-plane properties of UD composites and, ultimately, increased delamination resistance. With this in mind it is not surprising that the first experimental works focus on the effect of stitching and on how it can affect delamination resistance for NCF materials [62, 63, 64, 65]. The mechanical properties of NCF materials as a function of fabric weight are investigated in [66]. Two different lay-ups are examined, biaxial +45/-45 and quasi-isotropic 0/+45/90/-45 and the results are compared to those of alternative composite forms. Both layup configurations prove to possess superior mechanical response to woven roving laminates. Also it is indicated that the nature of the stitching or knitting yarn in the fabric plays a role in controlling the deformation and fracture of the final composite. The damage patterns of NCF materials are more recently examined in [67]. The damage caused during tension is labelled into four categories: longitudinal cracks, half cracks, whole cracks and double cracks. The most important finding is ‘longitudinal cracks’ which do not occur in UD composites when subjected to tensile loading conditions. This type of crack, depicted in Figure 14, is shown to be located within the bundles and seldom penetrated the matrix. Yet, at high strains cracks are found to grow through the matrix connecting to neighboring bundles. Mattson et al. [68] investigate the damage of NCF materials considering the effect of stacking sequence. They conclude that NCF composites are
very much dependent on the fabric layer stacking sequence for the case of tensile loading. Some of the latest experimental [69, 70] works indicate that the research of the NCF materials is still on-going.

The first papers on FE modelling of NCF materials can be traced back to the early 1990s. In 1994 FE modelling studies on stitched composites [71] revealed that stress gradients exist around the stitches due to differences on the thermal coefficients of the thread and composites. The compressive properties of NCF materials are investigated in [72]. The authors use a two-dimensional repeating FE model through the thickness of a biaxial fabric, to relate the NCF compressive properties to the geometrical and mechanical characteristics of the constituents. It is found that the compressive behavior of the NCF is controlled by 0° tows geometrical instability arising at the mesoscopic scale and accompanied by resin shear plastic flow. To improve the compressive performance the authors suggest several ways to reduce the crimp of

![Figure 14: Micrograph of the longitudinal cracks in an NCF cross-ply laminate [67]](image)

the tows which plays a critical role on the overall compressive behavior of the NCF. In [73] FE damage analyses is performed in order to study the progressive failure of [0/90]s non-crimp fabric composite laminates in tension. In order to model the damage onset and propagation stress based failure criteria, associated with a point-wise stiffness degradation scheme, are applied to the model through a user-defined material subroutine interfaced with ABAQUS. It is
suggested to use the Maximum Stress criterion for the tows and the Maximum Stress criterion or the Drucker–Prager model for the resin pockets.

Perhaps the most time-consuming modelling procedure of the woven materials is that of developing FE models considering the material nonlinearities, such as crimp of yarns. Because modelling of the entire geometry of a specimen is prohibitive due to time limitations, a very common technique is the development of RVE or UC. An RVE is the smallest possible volume to represent the properties of the macro-scale material and depends upon the material’s geometry. To aid the creation of such models, Lomov et al. [74, 75, 76, 77, 78] have developed the geometrical preprocessor Wise-Tex. Wise-Tex contains a textile pre-processor using the principle of minimum energy to calculate cross-section shapes and trajectories of the yarns. On the other hand Tex-Gen geometrical preprocessor was created by Sherburn [79]. In Tex-Gen software each yarn is created by defining the location of points called nodes and connecting those using splines. Both packages provide a great aid in developing RVE of material with complex geometries and are used by many researchers.

From the overview of the literature about NCF composite materials the following conclusions can be drawn:

- Lack of integrated methodologies that combine detailed FE models and simulation of the full mechanical response of NCF woven composites

### 2.4.2. 3D FIW COMPOSITE MATERIALS

Three-dimensional (3D)-fabrics were developed in the 1970’s [80], and are characterised by yarns oriented not only in-plane, but also in the through-thickness direction resulting in higher through-thickness strength and stiffness of the final composite material. 2D weaving machines were initially used for the production of 3D woven materials resulting in the angle interlocked woven fabrics. Later [81] as the 3D weaving process developed, new material configurations were able to be constructed like orthogonally interlocked fabrics. A promising solution to the demand for increasing matrix-dominated mechanical properties of composite
materials is the recently developed [81, 82, 83, 84] 3D FIW composite, represented in Figure 15, which contains yarns in the longitudinal (warp or W), transverse (horizontal weft or H-weft) and normal (vertical weft or V-weft) directions. In this architecture, interlace between all yarns takes place contrary to orthogonally or angle interlocked 3D fabrics where no enmeshing takes among different yarns. Schematic representations of different material setups of 3D woven composites are given in Figure 16.

Figure 15: 3D FIW material a) actual material, b) explanation of the material’s structure, c) YZ view and d) isometric view

The beginning for the experimental and theoretical study of 3D woven fabrics can be traced back more than two decades ago in the work of Brandt et al. [86]. Since then, a
A considerable number of experimental and numerical studies have been published [87, 88, 89, 90, 91, 92, 93]. Some recent studies in the area of numerical analysis are [94, 95, 96, 97, 98]. The majority of studies on 3D woven fabrics refers to orthogonally or angle-interlocked architectures. However, as 3D woven materials of different configurations have different mechanical properties and are governed by different failure mechanisms, it is impossible to draw conclusions for a new material architecture from existing data. To date, some of the mechanical properties of 3D FIW composites have been measured experimentally by Stig and Hallstrom [99].

Besides this, the

Figure 16: Different configurations of 3D woven materials. a) Angle interlocked, b) orthogonally interlocked, c) fully interlocked. Taken from [85]
elastic properties of the 3D FIW composites are evaluated numerically in [85, 100, 101]. Specifically, a step-by-step procedure for defining and creating an RVE from the nominal geometry of the 3D FIW composite is presented in [100]. A comparison between the cross sections of the FE model and the actual material has shown that this methodology can be effectively used for the development of RVEs of 3D FIW composite materials. Using the methodology developed by Stig and Hallstrom, FE models are created and used to compute stiffness of the material and study the effect of crimp on its elastic properties [100] [101]. The numerical studies of Stig and Hallstrom do not include failure and strength properties of the 3D woven material. Apart from the works Stig and Hallstrom, to the authors’ knowledge, no other work has been reported on the modelling of the mechanical behaviour of 3D FIW composites.

From the overview of the literature about NCF composite materials the following conclusions can be drawn:

- As in the case of woven materials no methodologies are found in the literature that combine detailed FE models and simulation of the full mechanical response of 3D FIW composites
- 3D FIW are very new materials, thus their mechanical characterization is not yet complete
- Lack of detailed FE models for 3D FIW composite materials
In this small chapter, the basic terms found in the thesis are defined in order to help the reader understand the following chapters.

**Stress Index:** The stress index in an element is derived from the appropriate failure criteria.

**Critical Stress Index:** The largest stress index in one part or material. If the structure is composed of more than one parts or materials each one has its own critical stress index.

**Failure Criterion:** An equation which considers the stress applied to an element as well as its strength to determine if it has failed or not. Among the many different failure criteria are the maximum stress criterion and the Hashin-Type failure criteria.

**Catastrophic Failure:** The last point on a stress-strain or load-displacement curve.

**Objective Function:** The largest stress index of all parts or different materials. The stress index responsible for catastrophic failure of the structure.

**Failure Transition:** Failure transition occurs when a critical stress index possess higher value than the objective function.

**Progressive Damage Modelling:** A numerical concept for modelling the damage propagation in a finite element model. It consists of the stress analysis, failure analysis and material property degradation modules.
**Definition of Basic Terms**

**Representative Volume Element or Unit Cell**: The smallest volume of a material that its mechanical properties can represent those of the material in the macro scale.

**Shape optimization**: Variation of the shape of a structure to achieve the optimization objective while not violating any constrains.
CHAPTER 4

THE NUMERICAL APPROACH FOR THE
OPTIMIZATION OF COMPOSITE STRUCTURES

4.1. DESCRIPTION OF THE NUMERICAL APPROACH

In the present thesis a numerical approach for the SO [8] of composite structures with regard to strength is proposed. The main advantages of the approach are:

- Its ability to simultaneously maximize the ultimate load while reducing the weight of the structural part.
- Its ability to investigate failure transition, a common problem in complex composite structures.
- Combination of PDM and SO in an automatic sub-routine.
- Can be applied to any composite structure with the least of modifications.

As stated in Chapter 2, FEA is a useful tool to perform optimization of complex structures. In the present study it is used to calculate critical values upon which the candidate solutions are evaluated. However, simple stress analyses cannot provide information about damage onset and progression. For this reason, PDM is embodied in the numerical approach. Its utility lies in predicting the initiation and propagation of damage, a crucial point in evaluating the overall mechanical response of the structure, and to track any failure transitions that might occur.

The aim of SO is to vary the values of geometric parameters in order to minimize critical stresses that arise in specific areas of the structure and lead to failure, as predicted by the FEA. The flowchart of the suggested approach is presented in Figure 17.
Figure 17: Flowchart of the numerical approach for the optimization
The numerical approach for the optimization of composite structures

In greater detail, the numerical approach is divided into three main legs namely, PDM of the initial geometry, optimization process and, final verification. In the first leg the 3D FE model of the structure is developed and solved using the PDM concept. A necessary condition for the continuation of the operation is the corroboration of the credibility of the FE model through comparison between numerical and experimental results. In the case where the numerical results diverge significantly from the experimental ones, the automatic process is stopped and the user is redirected to the development of the FE model. The importance of these actions cannot be stressed enough, as they lay the foundation for a reliable optimization procedure. On the contrary, if the validation is successful the program continues to the leg of the optimization based on the numerical results of the initial PDM.

The objective of SO is to optimize the ultimate strength of the structure by minimizing the stress that leads to ultimate structural failure, under weight constraints. Thus, the first step of the second leg is the classification of stress indices based on their severity. The stress indices for all elements are calculated using appropriate failure criteria which consider the strength of the material as well as the calculated stresses. Upon completion of this step, the variable due for minimization in the optimization process is chosen. This quantity referred to as OF, is defined as the stress index responsible for catastrophic failure of the structure according to the PDM results of the initial geometry. Concurrently, reduction of weight is imposed as constraint for the choice of optimal topology given the fact that an aircraft structure must be as light as possible. Subsequently the population of candidate solutions is generated. Each design is then solved for a given displacement to calculate the value of the OF. When the optimization process is completed the non-feasible designs are discarded. For each design, failure transition is inspected. In the case where failure transition has occurred, the program is terminated. Otherwise, based on the available sets, the one which minimize the value of the OF is chosen.

The verification of the approach takes place in the third leg. The optimal geometry is solved using PDM. Then, its load-displacement curve is compared to the initial one to ensure the enhanced ultimate load before failure. If the predicted strength of the optimum geometry is greater than the strength of the initial geometry the methodology is satisfactorily terminated. Yet, although failure transition is prevented in the second leg, in engineering problems with
The numerical approach for the optimization of composite structures

large interaction among different materials and parts failure transitions can still occur in later points on the load-displacement curve. In such a case, the user is re-directed in the beginning for a more detailed review on the problem.

4.1.1. **PDM**

A classical PDM [102, 103] comprises the modules of stress analysis, failure analysis and material property degradation. These modules are modified according to the geometry of the structural part, the loading conditions, and the types of used materials. The flowchart of PDM is presented in Figure 18

![PDM Flowchart](image)

Figure 18: PDM flowchart [102, 103]
In the module of stress analysis, the FE model is solved and its stresses are calculated to be used as inputs in the part of failure analysis. The stress analysis of a composite material is a complex problem as it exhibits orthotropic mechanical behaviour. Contrary to metals, composites possess more the one failure mechanisms which can be divided in two categories, fibre and matrix dominated. One of the most significant matrix dominated failure mechanisms for composite materials is delamination. Delamination occurs when two adjacent layers detach. This failure is the result of high normal and interlaminar shear stresses especially on the free edges [104, 105]. Three dimensional stress states are even more pronounced in complex composite structures that incorporate not only more than one component, but also different materials. As a result, stress analysis must be performed based on highly detailed 3D FE models.

The module of failure analysis investigates failure initiation at an element basis. Failure analysis is performed using failure criteria. In the recent years many researches have focused on the development of failure theories that can precisely predict the onset and propagation of failure in composite materials. Theoretical analysis of failure is a tool for the prediction of the mechanical response of composite materials. Failure criteria are equations that consider the applied stress at an element and the strength of material to decide whether an element has failed or not. They are in general based on theories of strength or fracture mechanics and can be separated into two classes regarding their ability to distinguish among the different failure modes incorporated in the failure of composite materials. Failure criteria like maximum stress criterion or Hashin-type criteria can account for the different failure modes of a composite material. On the contrary, failure theories like the Hoffman criterion do not have this ability. A very detailed review of the available failure theories and their applications can be found in [106]. The choice of any failure criteria in this thesis is based on the following requirements:

- Simplicity of implementation implement within FE code
- Ability to distinguish different failure modes
- Proven ability to predict failure in the desired application
The numerical approach for the optimization of composite structures

When failure is predicted in an element, its modulus is reduced preventing it from carrying load in the respective direction. Modulus degradation is based on degradation rules that can be either sudden [107, 108, 109] or gradual [110]. The degradation rules represent a mathematical formulation of the effect of different failure modes in the load-carrying capacity of an element. In the later chapters of the thesis, according to the specific application, the proper failure criteria and degradation concept are selected and explained in detail.

4.1.2. **OPTIMIZATION MODULE**

4.1.2.1. **OPTIMIZATION VARIABLES**

The optimization process is conducted, based on the optimization module of the ANSYS FE code [111]. The module employs three types of variables that characterize the design process: the DVs, the SVs and the OF.

The DVs are independent variables, such as geometrical parameters, which vary inside a certain range to achieve the optimum design. Variation of the DVs has an effect both on SVs and the OF. The values of these variables are altered during the optimization process, within limits, to produce the population of candidate solutions. The vector of DVs is defined by:

\[
\mathbf{x} = (x_1x_2x_3 \ldots x_n)
\]  

(1)

DVs are subject to \( n \) constraints with upper and lower limits, that is,

\[
x_j \leq x_i \leq \bar{x}_i \quad (i = 1, 2, 3, \ldots, n)
\]  

(2)

where \( n \) is the number of DVs.

The dependent or SVs are response quantities that are functions of the DVs, like weight or stresses. A SV may have upper and lower limits but it can also be single-sided having only
The numerical approach for the optimization of composite structures

one limit. These limits provide additional design restrictions apart from the ones provides by
the limits of the DVs. The mathematical form of SVs is

\[ g_i(x) \leq \bar{g}_i \quad (i = 1,2,3,\ldots,m_1) \]  
\[ h_i \leq h_i(x) \quad (i = 1,2,3,\ldots,m_2) \]  
\[ w_i \leq w_i(x) \leq \bar{w}_i \quad (i = 1,2,3,\ldots,m_3) \]

where \( g_i, h_i \) and \( w_i \) are SVs containing the design, with underbars and overbars
representing lower and upper bounds, respectively, and \( m_1 + m_2 + m_3 \) is the number of SVs
constraints with various upper and lower limit values.

The OF presented in Eq.(6) is the quantity, function of the DVs, to be minimized by the
optimization process

\[ f = f(\boldsymbol{x}) \]  

In the numerical approach, developed in this work, the DVs correspond to the
dimensions of the structural part. The variation of dimensions is subjected to constraints
defined by industrial specifications. The SV is the weight of the part. As the methodology is
focused on the increase of strength, the weight is not to be minimized but only to not be
increased. The OF to be minimized is a stress value. It is noted that if different material systems
and parts are used in the structure, the greater stress index among all indexes is used as the OF.

4.1.2.2. **Optimal Design Selection**

To create the population of the candidate design sets, the random design generation
method, a sub-type of the sub-problem approximation, is used. The random design generation
The numerical approach for the optimization of composite structures is selected as it is easily configured to the user’s programming needs and provides reliable results. At the same time, it provides increased accuracy and reduced computational time. The procedure for the evaluation of design sets is presented in Figure 19.

![Flowchart of the optimal design selection procedure](image)

Figure 19: Flowchart of the optimal design selection procedure
Initially a random design set $i$ is generated and stress analysis is performed with a pre-defined displacement. A complex structure may include $n$ distinct parts one or more of which could be made of different materials. Thereby, apart from the value of the OF, critical stress indices of all different parts of the structure are calculated in order to ensure that no failure transition has taken place. Failure transition can be identified in the structure due to stress interactions among different parts. When stress drops in one area of the structure this is likely to cause stress increase in another. The condition under failure transition is observed is presented in Eq 7.

$$f_{MIN} \leq SI_n$$

(7)

where $SI_n$ is the maximum SI calculated for the $n$th part and $f_{MIN}$ the minimum value of the OF.

The actual meaning of the above equation is that the part or area of the structure that is responsible for catastrophic failure in the case of the initial geometry, has shifted. Now a SI in the $n$th part is greater than the OF defined by the PDM results of the initial geometry. As a direct result, if for a design set $i$ failure transition has occurred the program stops and the user re-considers the optimization variables. Otherwise the program proceeds to the comparison of the OF calculated for the $i$th set with the lowest OF calculated until that moment as stated in Eq 8

$$f_{MIN} \geq f_i$$

(8)

If the OF is minimized in $i$th design, then this design is set as the optimal one. This process is repeated for the desired number of total design sets ($i+1$, $i+2$ etc.). Finally, the optimal design chosen based on the restrictions of Eq.(3)-(5) and Eq. (7)-(8) and is described in Eq 9

$$f_{FINAL} = f_{MIN}$$

(9)
4.2. **INNOVATIONS**

In this chapter a novel numerical approach for the SO of composite structures is developed. The proposed approach is capable of maximizing the mechanical strength of a composite structure while at the same time reducing its weight. Equally important is its ability to track any failure transition that occurs during the optimization procedure. Its main features are:

- Automated numerical approach
- Ability to track failure transition
- The combination of SO and PDM to evaluate the optimal design
- The ability to be applied in any composite structures without heavy modifications
5.1. GEOMETRY AND MATERIALS

Joining elements have been recently developed, as an alternative to traditional mechanical fastening techniques, to joint components with the method of adhesive bonding. Reliability and weight reduction are key aspects that must be fulfilled in order to introduce this concept as a viable alternative to mechanical fastening. Since H-shaped joints are among the less studied joins, it is wise to explain the different areas of the joint. A typical H-joint is illustrated in Figure 20.

Figure 20: Schematic illustration of an H-shaped joining element and indication different areas
Application of the numerical approach to the H-shaped joining element

The geometry and dimensions of the H-shaped profile are shown in Figure 21. The overall initial dimensions of the specimen considered in this work are presented in Figure 22.

Figure 21: Dimensions in mm of the H-shaped joining element

The stacking sequence of the NCF and 3D FIW composite materials are defined in Figure 23. For both novel composite materials there weren’t available material properties in the literature. Both materials contain geometrical nonlinearities which make their direct modelling in a non-specialized FE package a laborious task. Moreover, contrary to the NCF material, the 3D FIW material could not be modelled using techniques developed for UD materials. In order to overcome both problems a mesomechanical approach for simulating the mechanical performance of woven fabric structural parts developed in [52] is also used. For the FEM of both materials a methodology is developed in Chapter 5.
The plates which are connected by the joining element are fabricated from 0/90 NCF material. For the adhesive, the EA 9695 film [54] is used. The material properties of the adhesive are listed in Table 1.
Application of the numerical approach to the H-shaped joining element

![Figure 23: Stacking sequence of a) NCF material and b) 3D FIW material](image)

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>2548 MPa</td>
</tr>
<tr>
<td>Shear modulus</td>
<td>980 MPa</td>
</tr>
<tr>
<td>Tensile strength, $\sigma_{\text{max}}$</td>
<td>40 MPa</td>
</tr>
<tr>
<td>Shear strength, $\tau_{\text{max}}$</td>
<td>20 MPa</td>
</tr>
<tr>
<td>Mode I critical energy release rate, $G_{IC}$</td>
<td>0.5 N/mm</td>
</tr>
<tr>
<td>Mode II critical energy release rate, $G_{II_C}$</td>
<td>1 N/mm</td>
</tr>
</tbody>
</table>

Table 1: Material properties of EA9695 film adhesive [54]

5.2. **FE MODEL OF THE JOINT**

The stress analysis of the composite structure is performed by the FE method. This is the only step of the optimization methodology that is entirely case dependent in the sense that for every new application, an entirely new FE model must be built from the beginning. Using ANSYS FE code a 3D model of the H-profile was developed. The model is parametric and
Application of the numerical approach to the H-shaped joining element provides the ability to the user to independently change its geometrical characteristics and its mesh density. To create the model, the ANSYS 3D Structural solid, SOLID 185 element type was used. It is defined by 8 nodes having three degrees of freedom, translations in the nodal, X, Y and Z directions. This element type is suitable for modelling 3D structures consisting of either isotropic or orthotropic materials. The geometry of the element type and stress output coordinates are illustrated in Figure 24

![Figure 24: Node position and stress output coordinates of SOLID 185 element type](image)

A typical mesh of the FE model along with indication of different components is illustrated in Figure 25.

For the H joining element made from NCF the FE mesh is selected such as to model one element per dual layer (dual element). The same FE mesh is used for H made from the 3D FIW based on the assumption that the interlock of the layers is applied at each element row. As can be seen in Figure 25, the mesh at the bottom of the H joining element is denser to account for the interactions among the different parts. On the other hand to reduce computational time as much as possible, two different mesh densities are used for the insert. The part of the insert that lies within the joint follows the same mesh density as the joint. A coarser mesh is selected for the insert outside the joining element as plays a minor role in the mechanical response; high accuracy is not necessary. Amongst different layers and different constituents perfect bonding is
Application of the numerical approach to the H-shaped joining element

Figure 25: Typical FE mesh of the FE model and identification of different parts

assumed. Thus, the nodes of the adjacent elements are fully merged. Detailed pictures of the various different parts are given in Figure 26. For the sake of briefness only half of the parts are presented.

Overall, the model consists of 191800 elements and 203235 nodes. The loading case investigated in this thesis is the tension of the joining element. Axial pull-out of the insert was modelled by constraining the nodes at one end of the insert and applying an incremental axial displacement at the nodes of the other end. Figure 27 demonstrates the FE model with its boundary conditions for the case of tension
Application of the numerical approach to the H-shaped joining element

Figure 26: Mesh details of the different parts: a) H-profile, b) adhesive, c) gusset fillers and resin areas and d) insert

Figure 27: Boundary conditions for tension of the joining element
5.3. **FAILURE ANALYSIS AND MATERIAL PROPERTY DEGRADATION**

5.3.1. COMPOSITE MATERIALS

In the PDM procedure, normal and shear stresses of each element are stored in every step. Then they are compared to the material’s strength in the corresponding direction. The results of the numerical characterization of the composite materials used to fabricate the joint are found in Chapter 5. The full stress strain curves will be used for failure analysis. However in the case of linear behaviour, failure analysis for the elements of the H element will be performed using the Maximum Stress criterion according to the following equations:

\[
\frac{\sigma_i}{\sigma_{\text{max}}} \geq 1 \quad (i = X, Y, Z) \tag{10}
\]

\[
\frac{\tau_i}{\tau_{\text{max}}} \geq 1 \quad (i = XY, XZ, YZ) \tag{11}
\]

in the above equations \(\sigma_i\) (\(i = X, Y, Z\)) symbolize the normal stresses and \(\tau_i\) (\(i = XY, XZ, YZ\)) the shear stresses in a loadstep. \(\sigma_{\text{max}}\) and \(\tau_{\text{max}}\) are the tensile and shear strength respectively.

Although MSC can be considered as a rather outdated choice, here it is necessarily used due of the dual elements. Once failure is predicted the moduli of the failed elements are reduced according to a sudden degradation concept presented in Table 2. The purpose of property degradation is to disable the failed element for carrying a load at a specific direction. This concept distinguishes two types of failures: catastrophic and non-catastrophic. Catastrophic failure is assumed when damage occurs in the X, Y directions denoted in Figure 22. In this case, all element moduli are set to zero, thus preventing it of carrying any type of load. Damage in all other directions is regarded non-catastrophic and only the relevant moduli are degraded.
Material direction | Degradation rules
--- | ---
X | \( E_x = E_y = E_z = G_{xy} = G_{yz} = G_{xz} = 0 \)
Y | \( E_x = E_y = E_z = G_{xy} = G_{yz} = G_{xz} = 0 \)
Z | \( E_z = G_{xz} = G_{yz} = 0 \)
XY | \( G_{xy} = 0 \)
XZ | \( G_{xz} = 0 \)
YZ | \( G_{yz} = 0 \)

Table 2: Sudden property degradation concept

### 5.3.2. Adhesive

To predict damage in adhesives two of the most common methods are CZM and CDM. When CZM is used, interface elements are placed to simulate the debonding between the adherents. A constitutive equation is used to relate the tractions, \( \tau \), to the relative displacements, \( \delta \), at the interface and then a softening law is applied. The main disadvantage of this approach is the requirement for calibration of the input parameters needed due to deviation of the theoretical from the actual material properties of the adhesive. Besides, there are convergence difficulties arising during the Newton-Raphson non-linear solution of the softening process, which mainly occur under mode-II (sliding) loading conditions. Alternatively to the CZM approach, the CDM method has been proposed. Specific advantages compared to the CZM technique such as smaller requirements in computational effort and the absence of non-linear solution, are very important when the method is combined with a heavy post-processing analysis. This approach implements exactly the same theory with the CZM but without any convergence difficulties since it controls the softening process by modifying the stiffness of the adhesive elements. The constitutive equation in this case is used to relate the normal (shear) stresses, \( \sigma (\tau) \), to normal (shear) strains, \( \epsilon (\gamma) \). Different softening concepts are depicted in Figure 28.
Nevertheless to use CDM in a complex bonded structure such as the H-profile, its reliability and applicability must be assessed. To compare the performance of the two modelling approaches, the mode-I fracture behaviour of a DCB specimen is simulated. Initially the numerical results are compared against the CDM of [112]. Then numerical and experimental results from [113] are compared to validate the approach.

The CZM contained in ANSYS, implements the theory of Xu and Needleman [114]. For describing the behaviour of the interface elements, a surface potential is adopted:

$$\varphi(\delta) = e \sigma_{max} \bar{\delta}_n [1 - (1 + \Delta_n) e^{-\Delta_n e^{-\Delta^2}}]$$

(12)

where $\sigma_{max}$ is the maximum normal traction and $\bar{\delta}_n$ is the normal separation across the interface when $\sigma_{max}$ is reached and $\delta_r$ equals zero.

$$\Delta_n = \frac{\delta_n}{\bar{\delta}_n}$$

(13)
\[ \Delta_t = \frac{\delta_t}{\bar{\delta}_t} \]  

(14)

\( \bar{\delta}_t \) is the shear separation across the interface when shear maximum traction is attained. The shear and normal displacements in a given step are symbolized with \( \delta_t \) and \( \delta_n \) respectively.

The traction is defined as:

\[ T_n = \frac{\partial \varphi(\delta)}{\partial \delta_n} \]  

(15)

\[ T_t = \frac{\partial \varphi(\delta)}{\partial \delta_t} \]  

(16)

From Eqs. (13) and (14), we get the normal and shear traction of the interface

\[ T_n = e\sigma_{\text{max}}\Delta_n e^{-\Delta_n e \Delta_t^2} \]  

(17)

\[ T_t = e\sigma_{\text{max}} \frac{\bar{\delta}_n}{\delta_t} \Delta_t (1 + \Delta_n) e^{-\Delta_n e \Delta_t^2} \]  

(18)

The work of normal and shear separation is

\[ \varphi_n = e\sigma_{\text{max}} \bar{\delta}_n \]  

(19)

\[ \varphi_t = \sqrt{2}e\tau_{\text{max}} \bar{\delta}_t \]  

(20)

For the 3D stress state, the shear or tangential separations and the tractions have two components, \( \delta_{\text{t1}} \) and \( \delta_{\text{t2}} \) in the element’s tangential plane and one obtains:
Application of the numerical approach to the H-shaped joining element

\[ \delta_t = \sqrt{\delta_{t1}^2 + \delta_{t2}^2} \]  \hspace{1cm} (21)

The traction is then defined as:

\[ T_{t1} = \frac{\partial \varphi(\delta)}{\partial \delta_{t1}} \]  \hspace{1cm} (22)

\[ T_{t2} = \frac{\partial \varphi(\delta)}{\partial \delta_{t2}} \]  \hspace{1cm} (23)

The CDM implemented here is the one proposed by Moura and Chousal [112]. It is based on a linear softening law that correlates adhesive failure (debonding) with stiffness degradation. The stress-strain curve of the CDM concept is presented in Figure 29.

![Figure 29: Softening stress-strain curve of the CDM [112]](image)

Initiation of debonding at the peak of the bilinear graph, is detected using Maximum Stress criterion. After debonding onset, a damage parameter \( d \) is introduced to gradually reduce the elements’ modulus. Young’s \( E \) and shear modulus \( G \) of the elements at each load step, in relation with their initial values \( E_0 \) and \( G_0 \), are derived by:
Application of the numerical approach to the H-shaped joining element

\[ E_i = (1 - d_{\varepsilon,i})E_0 \]  
\[ G_i = (1 - d_{\gamma,i})G_0 \]  

The scalar parameter \( d_i \) of element \( i \) varies from 0 (undamaged state denoted by index \( n \)) to 1 (complete debonding denoted by index \( u \)) and is derived, for normal and shear strains respectively, from

\[ d_{\varepsilon,i} = \frac{\varepsilon_{ui}(\varepsilon_i - \varepsilon_{ni})}{\varepsilon_i(\varepsilon_{ui} - \varepsilon_{ni})} \]  
\[ d_{\gamma,i} = \frac{\gamma_{ui}(\gamma_i - \gamma_{ni})}{\gamma_i(\gamma_{ui} - \gamma_{ni})} \]

The normal and shear strains at debonding completion are calculated using the critical strain energy release rate, \( G_C \) in mode I and II, respectively, from:

\[ G_{IC} = \frac{1}{2} \sigma_{max} \varepsilon_{ui} l_c \]  
\[ G_{IIc} = \frac{1}{2} \tau_{max} \gamma_{ui} l_c \]

where \( l_c \) is a characteristic length transforming displacement to strain derived from:

\[ l_c = \sqrt[3]{abc} \]

where \( a, b, c \) are the dimensions of the solid element.
To create the 3D FE model of the DCB specimen ANSYS FE code is used. The adherents were modelled using the SOLID 185 element type which also has a layered option. In the case of [112] the adherents are metallic while, in [113] the adherents are made from CFRP. The adhesive is represented with the INTER205 interface elements, for the CZM, and with the SOLID185 element for the CDM. The FE mesh of the DCB specimen along with the boundary conditions is shown in Figure 30(a)-(c)

![DCB specimen](image)

Figure 30: DCB specimen. a) FE model, b) FE model details and c) boundary conditions

The comparison between the CDM and CZM approaches in terms of load displacement curves can be seen in Figure 31. Excellent agreement is achieved between CZM model, the CDM model and the model of [112] for the total mode-I mechanical response. In Figure 32 the
numerical results are compared to the experimental results of [113]. The model predictions can be regarded as accurate because they are capable of capturing the most important aspect of the mode-I fracture behaviour of the composite bonded joint, which is the maximum load. Also the initial stiffness as well as the initial post-failure behaviour are predicted satisfactorily. Furthermore, the CZM is more time-consuming by 50% as it implements a non-linear Newton-Raphson solution, while the CDM implements a fully adjustable non-linear solution. Based on these findings the CDM is used in this thesis to model the mechanical response of the adhesive material.

![Comparison of the simulated load-displacement curves](image)

*Figure 31: Comparison of the simulated load-displacement curves*
5.4. **INNOVATIONS**

In Chapter 4 a direct comparison between a CZM and a CDM took place in terms of, computational cost, reliability and applicability. The results indicated the potential use of CDM concept in complex problems. The advantages it offers in comparison with CZM approach are:

- Extremely accurate prediction of debonding initiation and propagation, given the fact that fine mesh and small loadstep is provided
- Elimination of numerical instability and convergence problems
- Exceptionally reduced computational cost
CHAPTER 6

NUMERICAL CHARACTERIZATION OF WOVEN COMPOSITE MATERIALS

6.1. MODELLING OF WOVEN COMPOSITE MATERIALS

Undoubtedly, woven composites possess some desired characteristics such as higher out-of-plane properties when compared to UD composites. Yet, research is still on-going on novel material configurations, for replacement of traditional composites to be enabled in aerospace applications. In order to reduce research costs as much as possible, the industry has focus on reducing mechanical testing by introducing alternative characterization methods such as virtual experiments. Reliability in this context is a key requirement, thus trustworthy FE models must be developed in order to simulate the mechanical response of woven composites for use in engineering applications.

Unlike UD plies, woven composite materials are highly inhomogeneous. Woven materials consist of two distinct phases, the fibre tows (fibre yarns embedded in matrix) and areas with pure resin called resin-rich areas or resin pockets. Although these features must be considered when an FE model is developed, it is very difficult due to time and computing power limitations- to model the exact microstructure of a specimen made of woven composites. Alternatively, RVEs of the materials can be developed to simulate their mechanical response and use it as input to a large scale model.

Yet, commercial FE packages do not possess special features for the modelling of woven composites. Consequently, modelling the material’s geometric nonlinearities directly in an FE package is a tedious and time-consuming task with doubtful results. Therefore, a methodology, based on the meso-mechanical modelling [52], is presently developed for the simulation of woven composite material’s mechanical response. The flowchart of the methodology is
Numerical characterization of woven composite materials

presented in Figure 33. The methodology can be applied to any woven composite with small modifications in the geometrical pre-processing section.

![Figure 33: Flowchart of the modelling methodology](image)

Initially the RVE of the actual material is selected. The RVE must effectively account for the micro-structure of the woven material, in order to efficiently transfer the mechanical behaviour from the micro- to the macro-level. Then, the nominal geometry of the RVE is created in a geometric pre-processor. The required inputs include the nonlinearities of the material, the matrix and fiber material properties, and the dimensions of the RVE. The outcome of this processing is the detailed geometrical model. Afterwards, the geometrical model is translated into FE code to develop the final FE model. Next, the PDM concept is applied to the FE model and the results are obtained. The results contain stress-strain curves and damage maps for all normal (X, Y, Z directions) and shear (XY, XZ, YZ plains) loadings. Finally, the calculated
results, in terms of stiffnesses and strengths, can be inserted as inputs into large-scale FE models.

### 6.1.1. Geometrical Pre-processors

To create geometrical models two established geometrical pre-processors can be used, namely WiseTex and TexGen software.

WiseTex was developed by Lomov and Verpoest [74, 75, 76, 78, 77]. The software is capable of modelling the geometry of UD preforms, stitched preforms, woven fabrics and 2D braids. Although it offers also modelling possibilities for angle and orthogonally interlocked 3D fabrics, it does not give a 3D FIW fabric option. On the other hand, WiseTex is ideal for modelling NCF materials due to the numerous available options. Moreover, it offers the possibility to define fiber and yarn properties. To calculate cross-section shapes and trajectories of the yarns, WiseTex uses the principle of minimum energy. In Table 3 the available options for each application are listed. After the geometrical model is created, FETex is used to export the geometry from WiseTex to ANSYS in the form of FE code. FETex is a program of the WiseTex suite which is used to translate WiseTex Geometrical models into ANSYS parametric language. The model can then be used to perform any type of FEA.

<table>
<thead>
<tr>
<th>Application</th>
<th>Defined properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fiber</td>
<td>Linear density, Diameter, Density, Friction coefficient, Young’s modulus, Shear modulus, Poisson ratio</td>
</tr>
<tr>
<td>Yarn</td>
<td>Cross section, Filament type, Fiber type, Mechanical properties</td>
</tr>
<tr>
<td>NCF</td>
<td>Number and Orientation of layers, Fiber type, Volume fraction, Stitching pattern, Stitching yarn, Dimensions</td>
</tr>
</tbody>
</table>

Table 3: Defined properties for various applications in WiseTex package
Numerical characterization of woven composite materials

TexGen originates from the work of Robitaille et al. [115, 116] [117, 118]. The current version of TexGen was developed in [79, 119]. Using this software, 2D and 3D woven fabrics can be modelled. Additionally to the pre-defined modelling options, the program offers the possibility to create user-defined yarn paths, which is beneficial for complex geometries such as 3D woven composites. Each yarn is created by defining the location of points called nodes and connecting those using splines. Then the matrix box and the final dimensions of the RVE can be set. Since the program does not have the option of exporting the geometrical model to ANSYS, the output file must be first processed in ANSYS Workbench.

According to [79], each software offers its own advantages:

TexGen Advantages:
- Fewer restrictions are placed on the geometry of the fabrics that can be modelled.
- Free and open source software.
- The software is tested on Windows and Linux.
- Possible to export geometry directly in IGES and STEP file formats.

WiseTex Advantages:
- Geometry calculation is based on physical properties of fabrics using analytical models
- Graphical user interface for creating a wider range of different fabric types.
- Built-in analytical models for the prediction of mechanical properties
- Ability to model gaps created through tows during stitching
6.2. **NCF COMPOSITE MATERIAL**

6.2.1. **GEOMETRICAL MODELLING**

To define the proper RVE of the NCF material, its structure is investigated. The RVE is chosen as the structural unit of the material that can represent the actual material. In Figure 34 the NCF material and the RVE definition are presented.

![Defined RVE](image)

**Figure 34: NCF RVE selection**

To create the geometrical model of the RVE of the NCF material, the WiseTex software is used due to the modelling options offered. The procedure followed to construct the geometrical models is described in Figure 35.

In the WiseTex pre-processor the required parameters needed for defining the geometry of the RVE, are the tow properties, the stitching properties and the final RVE dimensions. A typical WiseTex input window for NCF materials is shown in Figure 36.
Numerical characterization of woven composite materials

Figure 35: Flowchart of the geometrical modelling of the NCF material

Figure 36: WiseTex input window. In the upper part the properties of the tows are defined. In the lower part the definition of the stitching properties takes place

To fully define tow characteristics, the fibre type of the tow, the volume fraction, the number of the layers, each layer’s thickness and layer orientation must be defined. The tows in
the NCF material consist of Toho Tenax F-13 fibres. The material and mechanical properties of the fiber are recorded in Table 4. As the layup of the NCF material used in the H profile consists of 0/90 and 45/-45 layers, two RVEs are constructed to represent each stacking sequence. For the sake of briefness the 0/90 RVE is subsequently referred to as RVE 1 and the 45/-45 RVE as RVE 2. In Table 5 the layer properties of the two RVEs are listed. The stitching yarn properties consist of the material and the path of the yarn. The yarn is made of polyester and its path is ‘tricot loop’ type.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear density (tex)</td>
<td>0.066</td>
</tr>
<tr>
<td>Diameter (mm)</td>
<td>0.006909</td>
</tr>
<tr>
<td>Density (g/cm³)</td>
<td>1.76</td>
</tr>
<tr>
<td>Friction Coefficient</td>
<td>0.1</td>
</tr>
<tr>
<td>Young modulus (GPa)</td>
<td>239</td>
</tr>
<tr>
<td>Poisson Ratio</td>
<td>0.19</td>
</tr>
<tr>
<td>Shear Modulus (GPa)</td>
<td>100.4</td>
</tr>
<tr>
<td>Strength (MPa)</td>
<td>4620</td>
</tr>
<tr>
<td>Strain (%)</td>
<td>1.8</td>
</tr>
</tbody>
</table>

Table 4: Material and mechanical properties of the fiber Toho Tenax F-13

<table>
<thead>
<tr>
<th></th>
<th>RVE 1</th>
<th>RVE 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Layers</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Stacking Sequence</td>
<td>0/90</td>
<td>45/-45</td>
</tr>
<tr>
<td>Thickness of Layer</td>
<td>0.14mm</td>
<td>0.14mm</td>
</tr>
<tr>
<td>Volume Fraction %</td>
<td>55</td>
<td>55</td>
</tr>
<tr>
<td>Dimensions</td>
<td>7.5 × 4.5 × 0.373mm</td>
<td>7.5 × 4.5 × 0.373mm</td>
</tr>
</tbody>
</table>

Table 5: Layer properties of the NCF RVEs
The geometrical models along with RVE 1 and RVE 2 details, are shown in Figure 37 and Figure 38, respectively. Stitching causes deviations of fibres in a ply from their uniform directions. These deviations produce fibre-free zones near stitching locations which are called resin pockets or resin-rich areas. When the fibre-free zones stay located are called cracks, or they can form continuous channels in the ply. Cracks can be witnessed in the 90, 45, -45 layers while channels are observed only in the 0 ply. These nonlinearities play a very important for the mechanical response and damage pattern in NCF materials.

Figure 37: a) Geometrical model of the RVE 1, b) stitching yarn, c) 0 layer, d) 90 layer
6.2.2. FE MODELLING OF THE NCF RVE

To develop the 3D FE model, the FE-Tex software is used for the discretization of the geometrical model created in the Wise-Tex software. The output of FE-Tex code is a macro-routine containing the necessary commands for the development of the micro-mechanical RVE in ANSYS FE code. A typical input window of the FETex is shown in Figure 39. The FE models of RVE 1 and RVE 2 are presented in Figure 40.

Both tows and resin pockets were modelled using SOLID 45 3D structural solid element type. Each element is defined by 8 nodes which have three degrees of freedom, translations in the nodal, X, Y and Z directions. The specific element type can adopt either orthotropic or isotropic properties. The fiber/matrix system is Toho Tenax F-13 HTS/RTM6. Anisotropic material properties are assigned to the fibre tows treating them as UD plies, while isotropic properties are given to the resin areas. All values of the material properties are given in Table 6 and Table 7. In all models perfect bonding is assumed between the components thus the adjacent nodes are fully merged. Each FE model consists of 62224 elements and 22079 nodes.
Numerical characterization of woven composite materials

Figure 39: FETex input window. In the upper part the periodic boundary conditions are inserted. At the bottom, the matrix material properties are defined

<table>
<thead>
<tr>
<th>Property</th>
<th>Value (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal modulus ($E_x$)</td>
<td>109000</td>
</tr>
<tr>
<td>Transverse modulus ($E_y = E_z$)</td>
<td>8570</td>
</tr>
<tr>
<td>In-plane shear modulus ($G_{xy}$)</td>
<td>3300</td>
</tr>
<tr>
<td>Out-of-plane shear modulus ($G_{xz} = G_{yz}$)</td>
<td>3300</td>
</tr>
<tr>
<td>Longitudinal tensile Strength ($X_T$)</td>
<td>1700</td>
</tr>
<tr>
<td>Longitudinal compressive strength ($X_C$)</td>
<td>1378</td>
</tr>
<tr>
<td>Transverse tensile Strength ($Y_T = Z_T$)</td>
<td>67</td>
</tr>
<tr>
<td>Transverse compressive strength ($Y_C = Z_C$)</td>
<td>240</td>
</tr>
<tr>
<td>In-plane shear strength ($S_{xy} = S_{xz}$)</td>
<td>61</td>
</tr>
<tr>
<td>Out-of-plane shear strength ($S_{yz}$)</td>
<td>46</td>
</tr>
</tbody>
</table>

Table 6: Toho Tenal HTS/RTM6 ($V_t=55\%$) material properties
Numerical characterization of woven composite materials

<table>
<thead>
<tr>
<th>Property</th>
<th>Value (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>3955</td>
</tr>
<tr>
<td>Shear modulus</td>
<td>1000</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>69</td>
</tr>
<tr>
<td>Compressive strength</td>
<td>120</td>
</tr>
<tr>
<td>Shear strength</td>
<td>55</td>
</tr>
</tbody>
</table>

Table 7: RTM 6 material properties

Figure 40: FE model of the two RVEs a) 0 layer, b) 90 layer, c) 45 layer
6.2.3. **FAILURE ANALYSIS AND MATERIAL PROPERTY DEGRADATION**

6.2.3.1. **Tows**

Although many different failure criteria have been established for UD composites, there are no specific criteria for NCF materials. In [52] it is concluded that failure criteria for UD plies can be used effectively in the failure analysis of the tows. Failure analysis is performed using 3D stress-based failure criteria. Hashin-type criteria [120] are chosen as they are easily implemented in FE code and are capable of distinguishing between different failure modes. In addition they have been used successfully in [52, 121] to predict failure onset in tows of woven materials. The selected criteria can account for matrix cracking, fiber failure and delamination in tension and compression. The mathematical expressions of the failure criteria are as follows:

Matrix tensile cracking ($\sigma_{yy} > 0$)

$$\left( \frac{\sigma_y}{S_{YT}} \right)^2 + \left( \frac{\tau_{xy}}{T_{XY}} \right)^2 + \left( \frac{\tau_{yz}}{T_{YZ}} \right)^2 \geq 1 \quad (31)$$

Matrix compressive cracking ($\sigma_{yy} < 0$)

$$\left( \frac{\sigma_y}{S_{YC}} \right)^2 + \left( \frac{\tau_{xy}}{T_{XY}} \right)^2 + \left( \frac{\tau_{yz}}{T_{YZ}} \right)^2 \geq 1 \quad (32)$$

Fiber tensile failure ($\sigma_{xx} > 0$)

$$\left( \frac{\sigma_x}{S_{XT}} \right)^2 + \left( \frac{\tau_{xy}}{T_{XY}} \right)^2 + \left( \frac{\tau_{xz}}{T_{XZ}} \right)^2 \geq 1 \quad (33)$$
Fiber compressive failure \( (\sigma_{xx} < 0) \)

\[
\left( \frac{\sigma_x}{S_{XC}} \right)^2 \geq 1
\]  

(34)

Fiber-matrix shear-out \( (\sigma_{xx} < 0) \)

\[
\left( \frac{\sigma_x}{S_{XC}} \right)^2 + \left( \frac{\tau_{xy}}{T_{XY}} \right)^2 + \left( \frac{\tau_{xz}}{T_{XZ}} \right)^2 \geq 1
\]  

(35)

Delamination in tension \( (\sigma_{zz} > 0) \)

\[
\left( \frac{\sigma_z}{S_{ZT}} \right)^2 + \left( \frac{\tau_{xz}}{T_{XZ}} \right)^2 + \left( \frac{\tau_{yz}}{T_{YZ}} \right)^2 \geq 1
\]  

(36)

Delamination in compression \( (\sigma_{zz} < 0) \)

\[
\left( \frac{\sigma_z}{S_{ZC}} \right)^2 + \left( \frac{\tau_{xz}}{T_{XZ}} \right)^2 + \left( \frac{\tau_{yz}}{T_{YZ}} \right)^2 \geq 1
\]  

(37)

where \( \sigma_i \) (i=X,Y,Z) are the normal stresses and \( \tau_{ij} \) (ij=XY, YZ, XZ) the shear stresses at a given step. \( S_i \) and \( S_C \) (i=X,Y,Z) are the tensile and compressive strengths respectively, while \( T_{ij} \) (ij=XY, YZ, XZ) is the shear strength.

The above criteria are used to simulate the failure modes that cause the actual material to fail. Since the definition of damage in composites is not straightforward as it is in metals, it is wise to explain the underlying physics of these equations that will offer a better understanding of the results. Composite materials, when mechanically loaded, display many local failures. In this thesis they are described as ‘damage’ or ‘failure’. Their development, accounted to the load increase, is referred as ‘damage accumulation’. In composite materials exhibit two main damage mechanisms namely the fiber damage mechanism and the matrix one. Fiber failure in tension occurs when, due to excessive load in the longitudinal direction, a fiber is separated macroscopically. Fiber failure in compression accounts for the fiber buckling under compressive loading in the longitudinal direction. Matrix failure is divided in two categories, the transverse
Numerical characterization of woven composite materials

and the normal one. In both cases matrix cracks are formed that travel parallel to the fibres direction. In the case of transverse matrix failure matrix cracks are formed due to increase in the transverse load. On the other hand, when normal load exceeds a critical value the plies of the composite material start to detach forming what is called delamination.

Failure at the elements is investigated in every step of the PDM. When a failure criterion is satisfied, a material degradation concept is applied in order to simulate the damage in terms of stiffness reduction. Likely to the case of failure criteria, no material property degradation rules have been established purely for NCF composites. In this work the sudden degradation concept of Blackletter [109] is adopted which is presented in Table 8. The stiffness reduction virtually means that the element is unable to carry any load in the specific direction. This concept discriminates two types of failure, catastrophic and non-catastrophic. In catastrophic failure modes, like longitudinal tension, the elastic properties are reduced disabling the element from carrying any load. On the contrary, in non-catastrophic failure modes only the corresponding property is degraded

<table>
<thead>
<tr>
<th>Failure Mode</th>
<th>Ex</th>
<th>Ey</th>
<th>Ez</th>
<th>Gxy</th>
<th>Gyz</th>
<th>Gxz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Tension</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
</tr>
<tr>
<td>Longitudinal Compression</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
</tr>
<tr>
<td>Transverse Tension</td>
<td>1.00</td>
<td>0.01</td>
<td>1.00</td>
<td>0.2</td>
<td>0.01</td>
<td>0.2</td>
</tr>
<tr>
<td>Transverse Compression</td>
<td>1.00</td>
<td>0.01</td>
<td>1.00</td>
<td>0.2</td>
<td>0.01</td>
<td>0.2</td>
</tr>
<tr>
<td>In-plane shear XY</td>
<td>1.00</td>
<td>0.01</td>
<td>1.00</td>
<td>0.01</td>
<td>1.00</td>
<td>0.01</td>
</tr>
<tr>
<td>Out-of-plane shear XZ</td>
<td>1.00</td>
<td>1.00</td>
<td>0.01</td>
<td>1.00</td>
<td>1.00</td>
<td>0.01</td>
</tr>
<tr>
<td>Out-of-plane shear YZ</td>
<td>1.00</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
</tr>
<tr>
<td>Normal tension</td>
<td>1.00</td>
<td>1.00</td>
<td>0.01</td>
<td>0.2</td>
<td>1.00</td>
<td>0.2</td>
</tr>
<tr>
<td>Normal Compression</td>
<td>1.00</td>
<td>1.00</td>
<td>0.01</td>
<td>0.2</td>
<td>1.00</td>
<td>0.2</td>
</tr>
</tbody>
</table>

Table 8: Degradation Rules
6.2.3.2. **MATRIX**

The matrix plays a crucial role on the mechanical response of the RVE due to existence of resin pockets of considerable volume. Thus, a model that can accurately simulate the non-linear response of the matrix is incorporated into the PDM. To model the mechanical response of the matrix in tension, compression and shear, the multi-linear isotropic hardening material model developed by Rolfes et al [122] is used in the PDM. To implement the material model, the stress–strain curves of the matrix, shown in Figure 41, are used as input into the FE model in the form of tabulated data. A direct advantage of this method is that there is no curve fitting and, as a result, no parameter estimation. After the last point of the tabulated data (maximum stress in the curve), a sudden degradation of the material is applied by fully degrading all elastic moduli.

![Stress–strain curves of the multi-linear isotropic material model](image)

**Figure 41:** Stress–strain curves of the multi-linear isotropic material model [122]

6.2.4. **BOUNDARY CONDITIONS**

In an actual NCF material an RVE would be embedded within the periodic structure. Consequently in order to produce results that are as close as possible to those of the material in the macro-scale the periodicity must be applied through proper boundary conditions.
Numerical characterization of woven composite materials

Periodicity is achieved using constraint equations which are applied in the nodes of opposite sides. The periodic boundary conditions for each load-case are presented by the following equations. For briefness, only the cases of longitudinal tension, longitudinal compression and in plane shear are presented in the following equation groups.

**Longitudinal Tension**

\[
\begin{align*}
    u_x(x = L, -L) &= \frac{u_x}{2} \\
    u_y(x = L, -L) &= 0 \\
    u_z(x = L, -L) &= 0 \\
    u_x(y = W, -W) &= 0 \\
    u_y(y = W, -W) &= 0 \\
    u_z(z = t, -t) &= 0
\end{align*}
\]

(38)

**Longitudinal compression**

\[
\begin{align*}
    u_x(x = L, -L) &= -\frac{u_x}{2} \\
    u_y(x = L, -L) &= 0 \\
    u_z(x = L, -L) &= 0 \\
    u_x(y = W, -W) &= 0 \\
    u_y(y = W, -W) &= 0 \\
    u_z(z = t, -t) &= 0
\end{align*}
\]

(39)

**In plane shear**

\[
\begin{align*}
    u_x(x = L, -L) &= 0 \\
    u_y(x = L, -L) &= \frac{u_{xy}}{2} \\
    u_z(x = L, -L) &= 0 \\
    u_x(y = W, -W) &= \frac{u_{xy}}{2} \\
    u_y(y = W, -W) &= 0 \\
    u_z(z = t, -t) &= 0
\end{align*}
\]

(40)

where \( u_i \ (i = X, Y, Z) \) is the displacement in the respective direction. L, W, t, are the length, the width and the thickness of the RVE respectively.
6.2.5. **RESULTS**

6.2.5.1. **VALIDATION**

The PDM of the NCF material is validated by performing a comparison between the numerical results and experimental results available from the literature. For this purpose the experimental results of [123] are used. The comparison of experimental and numerical results for the RVE 1 and RVE 2 are presented in Figure 42 and Figure 43 respectively.

![Comparison of experimental and numerical results](image)

**Figure 42:** Comparison between predicted and experimental behavior of RVE 1 in a) longitudinal tension and b) longitudinal compression.

The comparison indicates that in both cases the mechanical properties are predicted with very good accuracy. More specifically, in the case of longitudinal tension, the stiffness prediction does not deviate from the experimental value. For the same loading case, the predicted ultimate stress and ultimate strain are in very good agreement with the experimental results. Particularly, the FE model underestimates the ultimate longitudinal stress and strain by 3% which is an acceptable deviation. The compressive properties of the NCF material are slightly overestimated in the FE models. The difference between predicted and experimental
stiffness is less than 5% while ultimate stress and strain are overestimated by 3%. Both of these values lie within acceptable limits.

**Figure 43:** Comparison between predicted and experimental behavior (modulus) of RVE 2 in a) tension and b) in-plane shear

Due to the lack of complete stress-strain curves of 45/-45 NCF materials only the stiffness values are compared. The predicted in-plane shear stiffness shows excellent agreement with the experimental value. On the other hand a slight deviation, of about 5%, is observed in the case of longitudinal tension. Based on the above, it can be concluded that the PDM simulates very well the mechanical behavior of the NCF specimen and thus it can be adopted for the full characterization of the NCF material.

### 6.2.5.2. STRESS-STRAIN CURVES

The complete simulated mechanical behaviour of RVE 1 is presented in Figure 44. All predicted stress-strain curves demonstrate a linear behaviour. Exceptions in this rule are some very small regions near the fracture where non-linearity is observed due to damage.
accumulation. For the longitudinal and transverse cases the tensile strength surpasses the compressive one. This is expected because tension is a fiber-dominated failure type whereas compression is dominated by matrix failure. On the other hand, normal tensile strength is significantly lower than the compressive one. This is attributed to the superior mechanical response of the matrix in compression as normal loading cases are purely matrix dominated. The calculated mechanical properties of RVE1 are given in Table 9.

Figure 44: Stress–Strain curves of RVE1. a) Longitudinal tension and compression, b) transverse tension and compression, c) normal tension and compression and d) in- and out-of-plane shear.
Numerical characterization of woven composite materials

<table>
<thead>
<tr>
<th>Loading case</th>
<th>Stiffness (MPa)</th>
<th>Strength (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal tension</td>
<td>57000</td>
<td>832</td>
</tr>
<tr>
<td>Longitudinal compression</td>
<td>57000</td>
<td>627</td>
</tr>
<tr>
<td>Transverse tension</td>
<td>59100</td>
<td>691</td>
</tr>
<tr>
<td>Transverse compression</td>
<td>59100</td>
<td>510</td>
</tr>
<tr>
<td>Normal tension</td>
<td>7533</td>
<td>50</td>
</tr>
<tr>
<td>Normal compression</td>
<td>7533</td>
<td>168</td>
</tr>
<tr>
<td>In-plane shear YX</td>
<td>2000</td>
<td>40</td>
</tr>
<tr>
<td>Out-of-plane shear XZ</td>
<td>2300</td>
<td>39</td>
</tr>
<tr>
<td>Out-of-plane shear YZ</td>
<td>2300</td>
<td>39</td>
</tr>
</tbody>
</table>

Table 9: Calculated properties for RVE1

The complete simulated mechanical behaviour of RVE2 is presented in Figure 45. Longitudinal, transverse and normal cases exhibit linear behaviour except some nonlinearity near the fracture points. However, in contrast to the RVE1, higher compressive strengths are predicted. This occurs because, unlike RVE1, the mechanical response in all cases is matrix dominated. On the contrary, all shear cases exhibit non-linear behaviour. The calculated mechanical properties of RVE2 are given in Table 10.
Numerical characterization of woven composite materials

Figure 45: Stress–Strain curves of RVE2. a) Longitudinal tension and compression, b) transverse tension and compression, c) normal tension and compression and d) in- and out-of-plane shear

<table>
<thead>
<tr>
<th>Loading case</th>
<th>Stiffness (MPa)</th>
<th>Strength (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal tension</td>
<td>10300</td>
<td>84</td>
</tr>
<tr>
<td>Longitudinal compression</td>
<td>10300</td>
<td>94</td>
</tr>
<tr>
<td>Transverse tension</td>
<td>10600</td>
<td>82</td>
</tr>
<tr>
<td>Transverse compression</td>
<td>10600</td>
<td>89</td>
</tr>
<tr>
<td>Normal tension</td>
<td>7500</td>
<td>36</td>
</tr>
<tr>
<td>Normal compression</td>
<td>7500</td>
<td>130</td>
</tr>
<tr>
<td>In-plane shear YX</td>
<td>3000</td>
<td>45</td>
</tr>
<tr>
<td>Out-of-plane shear XZ</td>
<td>2360</td>
<td>34</td>
</tr>
<tr>
<td>Out-of-plane shear YZ</td>
<td>2360</td>
<td>34</td>
</tr>
</tbody>
</table>

Table 10: Calculated properties for RVE2

5.2.5.3. DAMAGE MAPS

The predicted failure modes for RVE1 are presented from Figure 46 to Figure 49. An explanation of the colours used to represent damage is given in Table 11. In longitudinal tension ultimate failure occurs due to damage accumulation in the fibres of the 0 layer. Also excessive
Numerical characterization of woven composite materials

damage occurs in the matrix of layer 90. In longitudinal compression, the dominant failure mode is fiber-matrix shear failure in the layer 0 while minor matrix failure is predicted at the layer 90

<table>
<thead>
<tr>
<th>Color</th>
<th>Failure mode</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Unfailed element</td>
</tr>
<tr>
<td></td>
<td>Fiber failure</td>
</tr>
<tr>
<td></td>
<td>Matrix failure</td>
</tr>
<tr>
<td></td>
<td>Delamination</td>
</tr>
<tr>
<td></td>
<td>Fiber-Matrix Shear</td>
</tr>
</tbody>
</table>

Table 11: Color pattern of failed elements

Figure 46: RVE1 longitudinal tension and compression failure modes at the final point of the respective stress-strain curve for the 0 and the 90 plies.
Like in the longitudinal tension, extensive failure of the fibers is the governing damage mode that leads to failure, along with some minor matrix damage. Moreover matrix failure is predicted in the elements of the layer 0. Again, similarly to longitudinal compression, ultimate failure of RVE1 in transverse compression is the result of fiber matrix-shear failure.

![Image of failure modes](image)

**Figure 47**: RVE1 transverse tension and compression failure modes at the final point of the respective stress-strain curve for the 0 and the 90 plies

Ultimate failure in both normal tension and normal compression takes place due to delamination failure in both 0 and 90 plies. Especially in normal compression minor shear damage is predicted.
Numerical characterization of woven composite materials

<table>
<thead>
<tr>
<th>0 layer</th>
<th>90 layer</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Tension</strong></td>
<td><strong>Compression</strong></td>
</tr>
<tr>
<td><img src="image1" alt="0 layer tension" /></td>
<td><img src="image2" alt="90 layer tension" /></td>
</tr>
<tr>
<td>Strain: 0.0075</td>
<td>Strain: 0.022</td>
</tr>
</tbody>
</table>

**Figure 48**: RVE1 normal tension and compression failure modes at the final point of the respective stress-strain curve for the 0 and the 90 plies

In in-plane shear loading, matrix failure in both 0 and 90 plies is the governing type of failure. On the other hand, more complex damage patterns are predicted for the out-of-plane shearing. There, a combination of fiber-matrix shear and matrix failure lead to catastrophic failure. Additionally, in the YZ case, delamination has occurred is some elements at the base of the layer 90.
Numerical characterization of woven composite materials

<table>
<thead>
<tr>
<th>0 layer</th>
<th>90 layer</th>
</tr>
</thead>
<tbody>
<tr>
<td>In-plane shear</td>
<td></td>
</tr>
</tbody>
</table>

| Strain: 0.017 |

<table>
<thead>
<tr>
<th>Out-of plane shear XZ</th>
</tr>
</thead>
</table>

| Strain: 0.017 |

<table>
<thead>
<tr>
<th>Out-of plane shear YZ</th>
</tr>
</thead>
</table>

| Strain: 0.018 |

Figure 49: RVE1 in- and out-of-plane shear failure modes at the final point of the respective stress-strain curve for the 0 and the 90 plies
The failure modes that appear in the transverse loadings are identical to the longitudinal cases due to material symmetry. The RVE failed mainly due to extensive damage in the matrix. Minor shear damage is observed in the compression case.

The predicted failure modes for RVE2 are presented from Figure 50 to Figure 53. Due to material symmetry only one ply is presented. In longitudinal tension and compression cases the main failure mode is matrix cracking. Yet, minor shear failure is predicted in compression.

![45 layer Tension and Compression](image1)

| Strain: 0.0085 | Strain 0.009 |

Figure 50: RVE2 longitudinal tension and compression failure modes at the final point of the respective stress-strain curve for the 45 ply

![45 layer Tension and Compression](image2)

| Strain: 0.0085 | Strain 0.009 |

Figure 51: RVE2 transverse tension and compression failure modes at the final point of the respective stress-strain curve for the 45 ply
Failure in normal tension and normal compression takes place due to extensive delamination between the plies.

In plane shear failure is caused by matrix damage accumulation. Out-of-plane ultimate failure is the result of the propagation of delamination.

**Figure 52**: RVE2 normal tension and compression failure modes at the final point of the respective stress-strain curve for the 45 ply.
6.3. **3D FIW Composite Material**

6.3.1. **Geometrical Modelling**

To simulate the mechanical response of the 3D FIW composite, a 3D model of the original geometry of the material is created. A schematic illustration of the nominal geometry of the 3D FIW material and the selection of the RVE are depicted in Figure 54. Due to lack of
information about the geometrical characteristics of 3D FIW materials all required data are taken from the works of Stig et. al [99, 100, 101, 85]

To create the geometrical model of the 3D FIW material RVE, the TexGen pre-processor is utilised. Conversely to the WiseTex, TexGen offers the ability not only to define random paths for the tows, but also to define custom cross-sections at any point of the tow. These modelling options give a great advantage in complex geometries such as 3D woven materials. The procedure followed is described in terms of flowchart, in Figure 55.

The starting point in creating a geometric model in the WiseTex pre-processor, is the definition of the coordinates of the points referred to as nodes. The nodes are critical points for

![Figure 54: Definition of RVE's geometry [100]](image)

![Figure 55: Flowchart of the geometrical modelling of the 3D FIW material](image)
both defining the boundaries of different tows as well as for designating any change in a tow’s
direction. Then splines, which correspond to the paths of the tows, are used to connect the
nodes. In order to simulate the crimp of the warp yarns, the Bezier spline is selected as it offers
the best representation of the path that the warp tows follow. On the contrary, crimp of the weft
yarns is considered non-existent [100]. Thus the nodes are connected with straight paths A
Bezier curve, shown in Figure 56, is defined by four point \( R_1, R_2, R_3 \) and \( R_4 \) in 3D space. The
parametric equation for a Bezier curve, \( B \), is defined as:

\[
B(x) = R_1(1 - x)^3 + 3R_2x(1 - x)^2 + 3R_3x^2(1 - x) + R_4x^3
\]

\[
0 \leq x \leq 1
\]

First, the rudimentary description of the paths that master tows follow takes place. The
master tows are found in the core of the RVE. Their paths are defined in Figure 57. In the actual
material all the tows are packed together. As a result, in the defined space of the RVE there is
penetration of adjacent tows. To model this effect and represent the periodicity of the real
material, additional tows (slave tows) are created. The slave tows are copies of the master tows
which are placed in space according to material microstructure. The final result is depicted in
Figure 58.
Figure 57: Definition of the node positions and Bezier splines that describe the tow’s paths

Figure 58: Definition of slave tows

After the rudimentary area description, cross sections of the tows must be assigned. This is the most important part of the modelling process as the cross sections of the yarns depend on
Numerical characterization of woven composite materials

the volume fraction of the tows within the RVE, \( v_s \), the average fibre volume fraction within the strands \( v_f^s \), and the overall volume fraction \( v_f \). These variables are connected through the equation:

\[
v_s = \frac{v_f}{v_f^s}
\]  

(42)

Special attention is given to finding the correct tow cross sections in order to achieve a global fibre volume fraction, \( v_f \) equal to 0.54, as specified in the material parameters of Stig and Hallstrom [99]. The value of all volume fractions are listed in Table 12. The final version of the geometrical model is depicted in Figure 59.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total volume fraction ( v_f )</td>
<td>54</td>
</tr>
<tr>
<td>Volume fraction within the tows ( v_f^s )</td>
<td>72</td>
</tr>
<tr>
<td>Volume fraction of the tows ( v_s )</td>
<td>75</td>
</tr>
</tbody>
</table>

Table 12: Values of volume fractions for the 3D FIW RVE

Due to limited amount of data on the geometrical modelling of the 3D FIW material, validation of the geometrical model is performed by comparing cross sections of the model to CT scans of the actual material found in [99]. The comparison, shown in Figure 60, indicates that good agreement is achieved.
Numerical characterization of woven composite materials

Figure 59: Geometrical model of the 3D FIW composite

Figure 60: Comparison between cross section of the actual material and geometrical model in a) in X-Z plane, b) in Y-Z plane. The images of the 3D FIW’s cross section are obtained by CT scans [99]
6.3.2.  FE MODELLING OF THE 3D FIW RVE

Stress analysis is performed based on the FE method. To this end, an FE model of the RVE is created using the ANSYS FE code. TEXGEN offers two options for exporting a geometrical model as an FE model: a dry fibre FE model and a voxel model. Initially, the option of dry fibres is chosen as it reflects the exact geometry of the 3D FIW composite. In this case, only the fibre tows are exported as shell tubes. Yet, the creation of matrix to enclose the tows leads to meshing problems and numerical instability due to irregular geometries. Consequently, the more convenient approach of a voxel model is selected. The voxel model consists of cubic shaped elements that are stacked to create the FE model. A typical voxel-based FE mesh of the RVE is shown in Figure 61. Both matrix and tows are modelled as solid entities and meshed using the ANSYS 3D 8-node SOLID185 element, which has three degrees of freedom per node. Between the tows and the matrix, a fully bonded feature was assumed; thus, the nodes of the adjacent elements were fully merged. Orthotropic and isotropic material properties are considered for the tows and matrix, respectively listed in Table 13 and Table 14.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal tensile modulus (E\textsubscript{XT})</td>
<td>155000</td>
</tr>
<tr>
<td>Longitudinal compressive modulus (E\textsubscript{XC})</td>
<td>117000</td>
</tr>
<tr>
<td>Transverse modulus (E\textsubscript{Y} = E\textsubscript{Z})</td>
<td>11500</td>
</tr>
<tr>
<td>In –plane shear modulus (G\textsubscript{XY})</td>
<td>4500</td>
</tr>
<tr>
<td>Out -of –plane shear modulus (G\textsubscript{XZ} = G\textsubscript{YZ})</td>
<td>4500</td>
</tr>
<tr>
<td>Longitudinal tensile Strength (X\textsubscript{T})</td>
<td>2800</td>
</tr>
<tr>
<td>Longitudinal compressive strength (X\textsubscript{C})</td>
<td>2000</td>
</tr>
<tr>
<td>Transverse tensile Strength (Y\textsubscript{T} = Z\textsubscript{T})</td>
<td>87</td>
</tr>
<tr>
<td>Transverse compressive Strength (Y\textsubscript{C} = Z\textsubscript{C})</td>
<td>290</td>
</tr>
<tr>
<td>In –plane shear strength (S\textsubscript{XY} = S\textsubscript{XZ})</td>
<td>90</td>
</tr>
<tr>
<td>Out -of –plane shear strength (S\textsubscript{YZ})</td>
<td>80</td>
</tr>
</tbody>
</table>

Table 13: Toray T700/ Reichhold Dion 9500 (V\textsubscript{i}=72%) material properties
Numerical characterization of woven composite materials

<table>
<thead>
<tr>
<th>Property</th>
<th>Value (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>3955</td>
</tr>
<tr>
<td>Shear modulus</td>
<td>1000</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>69</td>
</tr>
<tr>
<td>Compressive strength</td>
<td>120</td>
</tr>
<tr>
<td>Shear strength</td>
<td>55</td>
</tr>
</tbody>
</table>

Table 14: Reichhold Dion 9500 material properties

An important aspect of the FE model, also discussed in [96, 98, 124] is the handling of the element coordinate system in the tows where crimp occurs. Regarding the tension load case, in the crimped warp tows, the element coordinate system is assumed to be constant and parallel to the global coordinate system in an attempt to simulate the straightening occurring during tension. In the compression load case, in which the coordinate system deviates in large areas of the warp tows, the local coordinate system of the elements is rotated according to the orientation of the elements with regard to the global coordinate system of the RVE. The two different approaches are shown in Figure 62.

Figure 61: Voxel FE model of the 3D FIW composite material
6.3.3. FAILURE ANALYSIS AND MATERIAL PROPERTY DEGRADATION

6.3.3.1. Tows

The tows of the 3D FIW composite material are treated as (thick) UD plies. Failure in composite plies can be either fibre dominated (e.g. fibre breakage) or matrix dominated (e.g. transverse matrix cracking and delamination). Because of the interaction of the different failure modes and the fact that severity of each failure mode is a function of loading, it is very important for the failure analysis module to consider all basic failure modes. Same as in the NCF case, the Hashin-type failure criteria are used to predict the following failure modes: fibre failure in tension, transverse matrix cracking in tension and compression, and normal matrix cracking in tension and compression. The Maximum Stress criterion is used to predict fibre failure in compression. Hashin-type failure criteria have been presented in detail in Chapter 5.2.3.1.

Once failure is predicted at an element, its elastic moduli need to be degraded accordingly to simulate the effect of damage on its load-carrying capacity. In other studies,
Numerical characterization of woven composite materials

Sudden property degradation rules are applied to model post-failure element behaviour of UD plies [109, 120]. However, this approach seems to be ineffective for woven composites [125, 126] for which a gradual degradation scheme seems more appropriate. In the present work, the damage mechanics concept developed by Matzenmiller et al. [110] and successfully used in [127, 128] is used to gradually degrade the elastic moduli of the tows. According to the damage mechanics concept, the presence of damage is simulated by the reduction of stiffness realised through the following function:

\[ E_i = (1 - \omega_i)E_0 \]  \hspace{1cm} (43)

\[ G_i = (1 - \omega_i)G_0 \]  \hspace{1cm} (44)

\[ \omega_i = 1 - e^{\frac{1}{2} m(1-r_j^m)}, r_j > 1 \]  \hspace{1cm} (45)

where \( r_j \) is the value of corresponding the Hashin-type criterion. The reduction of the elastic moduli is governed by the damage parameter, \( \omega_i \) which varies exponentially according to Eq. (45). Damage parameter, \( m \), controls the softening response of the material. The stress–strain behaviour of the elements implied by the gradual degradation concept is schematically described in Figure 63. It can be seen that for smaller values of, \( m \), the material obtains a more ductile response, whereas higher values give the material more brittle behaviour. The value of the damage parameter \( m \), is chosen from a parametric study that is presented in subsequent results section.
6.3.3.2. **MATRIX**

As with NCF, resin pockets are formed due to the inhomogeneity of the 3D FIW material. The resin pockets play a major role on the overall mechanical response and especially in the out-of-plane loading cases. To successfully model the highly non-linear response of the matrix a suitable material model must be implemented. Similarly to the NCF case the elastic-plastic material model developed by Rolfes et al. [122] is used in the PDM. The model has been described in detail in Chapter 5.2.3.2.

### 6.3.4. **RESULTS**

#### 6.3.4.1. **VALIDATION**

To validate the PDM, a comparison with experimental results from the study of Stig and Hallstrom [99] is performed for the case of longitudinal tension and compression. Also, analytical results of the Mori-Tanaka theory, found in [100], are compared to the simulated results for the cases where experimental data are not available. Within the validation
framework, a mesh convergence study and a parametric study on the effect of damage parameter $m$ are also conducted. In the mesh convergence study, the criterion adopted is 5% deviation in the stress–strain curve between consecutive meshes. Figure 64 compares the numerical stress–strain curves, computed with three different FE meshes, with two experimental curves for the load case of longitudinal tension. The analyses are conducted using a displacement increment of 0.0075 mm, which is selected by taking into account results from a separate parametric study and the required computational effort. The computational times of the analyses with the meshes of 64 000, 128 000 and 256 000 elements are 4, 11 and 24 h, respectively, in a workstation with a dual core AMD Opteron Processor 2218 at 2.6GHz and 16GB RAM. As it may be seen, convergence is achieved between meshes with 128 000 and 256 000 elements. Consequently, all analyses are performed using the mesh with 128 000 elements.

To study the effect of $m$, analyses with $m=1$, $m=2$ and $m=4$ are performed with the mesh of 128 000 elements. The curves from the parametric study on $m$ are included in Figure 65. As it may be seen, the curve obtained with $m=2$ agrees very well with the experimental curves both with regard to stiffness, failure stress and failure strain. The analysis with $m=1$ overestimates failure stress and failure strain as it implies a softening part in the stress–strain curve with a smaller slope, whereas the analysis with $m=4$ underestimates failure stress and failure strain as it implies a softening part in the curve with a higher slope. Therefore, the value of 2 was chosen for $m$. 
Numerical characterization of woven composite materials

Figure 64: Comparison between predicted and experimental stress–strain curves for different mesh densities. Damage parameter \( m = 2 \) in all analyses.

Figure 65: Comparison between predicted and experimental stress–strain curves for different values of damage parameter \( m \). Number of elements equals to 128,000 in all analyses.
In the case of longitudinal compression, for which no experimental curves are available, the predicted and experimental Young’s moduli are 64.5 GPa and 68.7 ± 5.3 GPa, respectively, and the predicted and experimental normalised failure loads are 758 N/mm and 694 ± 177 N/mm, respectively. All other available numerical data [100] are compared to numerical results in Table 15. The comparison between numerical results and results found in the literature, is satisfactory.

<table>
<thead>
<tr>
<th>Property</th>
<th>Numerical Value</th>
<th>Analytical Value (Mori-Tanaka)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile transverse modulus</td>
<td>15.4 (GPa)</td>
<td>15.6 (GPa)</td>
</tr>
<tr>
<td>In-plane shear modulus</td>
<td>3.3 (GPa)</td>
<td>3.77 (GPa)</td>
</tr>
<tr>
<td>Out-of-plane shear modulus</td>
<td>3.2 (GPa)</td>
<td>2.07 (GPa)</td>
</tr>
</tbody>
</table>

Table 15: Comparison between predicted and numerical properties of the 3D FIW material

6.3.4.2. **STRESS-STRAIN CURVES**

The predicted stress–strain curves are gathered in Figure 66. In all cases, a linear material behaviour is predicted except for the regions close to final failure where the accumulated damage in the matrix and the tows caused a non-linear behaviour. The highest non-linearity is predicted for longitudinal tension, and it is caused by the accumulation of matrix cracking in the tows and the matrix of the RVE. The material’s response in transverse and normal directions is similar because of material symmetry. It is noteworthy that in both transverse and normal directions, the material has higher compressive than tensile ultimate stress, which is attributed to the better compressive response of the matrix.
Figure 66: Predicted Stress-Strain Curves of the 3D FIW material. a) Longitudinal tension and compression, b) transverse tension and compression, c) normal tension and compression and d) in- and out-of-plane shear

The values of stiffness and strength at each direction of the material are compared in Table 16 with the properties of the UD AS4/3501-6 ply [120] of similar fibre volume fraction. The properties for which the 3D FIW composite excels are indicated as with grey-shaded. These are the transverse and normal elastic moduli, the out-of-plane shear modulus, the transverse and normal tensile strengths, and the out-of-plane shear strength. This finding verifies that the
presence of fibres in the normal direction leads to enhancement of the out-of-plane properties of the 3D FIW composite material.

<table>
<thead>
<tr>
<th>Property</th>
<th>3D FIW</th>
<th>3501-6/AS4 UD</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{XT}$</td>
<td>70.4 GPa</td>
<td>147 GPa</td>
</tr>
<tr>
<td>$E_{XC}$</td>
<td>64.54 GPa</td>
<td>128 GPa</td>
</tr>
<tr>
<td>$E_{YT}=E_{ZT}$</td>
<td>15.4 GPa</td>
<td>9 GPa</td>
</tr>
<tr>
<td>$G_{XY}$</td>
<td>3.3 GPa</td>
<td>5 GPa</td>
</tr>
<tr>
<td>$G_{XZ}=G_{YZ}$</td>
<td>3.2 GPa</td>
<td>3 GPa</td>
</tr>
<tr>
<td>$X_T$</td>
<td>1003 MPa</td>
<td>2004 MPa</td>
</tr>
<tr>
<td>$X_C$</td>
<td>379 MPa</td>
<td>1197 MPa</td>
</tr>
<tr>
<td>$Y_T=Y_Z$</td>
<td>144.4 MPa</td>
<td>53 MPa</td>
</tr>
<tr>
<td>$Y_C=Z_C$</td>
<td>185 MPa</td>
<td>204 MPa</td>
</tr>
<tr>
<td>$S_{XY}$</td>
<td>60.7 MPa</td>
<td>137 MPa</td>
</tr>
<tr>
<td>$S_{XZ}=S_{YZ}$</td>
<td>54.5 MPa</td>
<td>42 MPa</td>
</tr>
</tbody>
</table>

Table 16: Comparison between predicted mechanical properties of the 3D FIW composite material and the properties of the UD AS4/3501-6 ply [129]

6.3.4.3. **Damage Maps**

The predicted failure modes for RVE1 are presented in Figure 67 to Figure 70. An explanation of the colours used to represent damage is given in Table 17

<table>
<thead>
<tr>
<th>Color</th>
<th>Matrix</th>
<th>Tows</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Unfailed element</td>
<td>Unfailed element</td>
</tr>
<tr>
<td></td>
<td>Failure due to shear stresses</td>
<td>Matrix failure</td>
</tr>
<tr>
<td></td>
<td>Failure due to normal stresses</td>
<td>Fiber failure</td>
</tr>
</tbody>
</table>

Table 17: Color pattern of failed elements
Ultimate failure in longitudinal tension is caused by extensive damage accumulation in the matrix of both pure matrix and tow elements. Additionally, fiber damage is predicted at the edges of warp tows. The same damage mode is responsible for failure in compression. Crimp of the warp tows plays a major role on the onset and propagation of matrix failure in the warp elements.

<table>
<thead>
<tr>
<th>Matrix</th>
<th>Warp Tows</th>
<th>Weft Tows</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Tension</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Strain: 2%</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Compression</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Strain 0.8%</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 67: 3D FIW longitudinal tension and compression failure modes at the final point of the respective stress-strain curve for the matrix, warp and weft tows.

Extensive matrix failure, especially in the pure matrix and warp elements, is also the governing damage type in the transverse tension case. On the contrary, fiber failure in the weft tows is responsible for catastrophic failure in transverse compression. Practically no failure is predicted in the warp tows.
The failure modes for normal tension and compression are identical to those of transverse cases due to material symmetry.

In both in-plane and out-of-shear loading cases matrix damage accumulation causes final failure of the material.
## Numerical characterization of woven composite materials

<table>
<thead>
<tr>
<th></th>
<th>Matrix</th>
<th>Warp Tows</th>
<th>Weft Tows</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tension</td>
<td></td>
<td><img src="image1" alt="Tension Matrix" /></td>
<td><img src="image2" alt="Tension Weft" /></td>
</tr>
<tr>
<td>Strain 1.4%</td>
<td><img src="image3" alt="Tension Warp" /></td>
<td><img src="image2" alt="Tension Weft" /></td>
<td></td>
</tr>
<tr>
<td>Compression</td>
<td><img src="image1" alt="Compression Matrix" /></td>
<td><img src="image2" alt="Compression Weft" /></td>
<td></td>
</tr>
<tr>
<td>Strain 1.3%</td>
<td><img src="image3" alt="Compression Warp" /></td>
<td><img src="image2" alt="Compression Weft" /></td>
<td></td>
</tr>
</tbody>
</table>

Figure 69: 3D FIW normal tension and compression failure modes at the final point of the respective stress-strain curve for the matrix, warp and weft tows
Numerical characterization of woven composite materials

<table>
<thead>
<tr>
<th>Matrix</th>
<th>Warp Tows</th>
<th>Weft Tows</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>In-plane shear</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Strain 2%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Out-of-plane shear XZ</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Strain 2%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Out-of-plane shear YZ</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Strain 2%</td>
<td></td>
</tr>
</tbody>
</table>

Figure 70: 3D FIW in- and out-of-plane shear failure modes at the final point of the respective stress-strain curve for the matrix, warp and weft tows

6.4. **INNOVATIONS**

Due to the lack of data on the mechanical properties of woven fabric composite materials a methodology for the numerical simulation of their mechanical response is
developed. The methodology combines highly detailed FEM with the PDM concept to predict the mechanical response. Its main features are:

- Geometrical pre-processing to create accurate FE models
- PDM is applied for full numerical characterization
- Applicability in any woven composite material without modifications

The methodology was used to simulate the mechanical behavior of a novel 3D FIW composite material. This is the first study in which full mechanical simulation of such materials is carried out.
CHAPTER 7

7. OPTIMIZATION RESULTS

7.1. JOINING ELEMENT FABRICATED OF NCF COMPOSITE MATERIAL

7.1.1. FE MODEL VALIDATION - INITIAL PDM

FE model validation is the most important step, as the optimization is unrealistic in case the PDM is not capable of accurately predicting the strength of the composite structure. Since there are no experimental data for the H joint fabricated of 3D FIW material, only the NCF material configuration is verified against the experimental results of [54]. The numerical and experimental load-displacement curves of the joint with the NCF H are compared in Figure 71.

Figure 71: Comparison between experimental and numerical results for the H joint fabricated of NCF material
Very good agreement is obtained regarding the overall mechanical behaviour of the joint in tension. The difference in ultimate load is less than 1% while the difference in ultimate displacement is less than 3%. The stiffness of the joint is underestimated by about 5% in the FE model. Equally important is capturing the primary failure mode as this will determine the OF in the optimization process. The model predicts that debonding between the H-joint and the insert is responsible for catastrophic failure as shown in Figure 72. An explanation of the colours used to represent damage is given in Table 18. Debonding initiates at the edges of the H at 0.875 mm and propagates towards the centre of the H joining element. Complete debonding is predicted at 1.4 mm. At that stage, delamination is also predicted at the base of H near the gusset filler areas. These findings coincide perfectly with the experimental results of [54].
<table>
<thead>
<tr>
<th>Adhesive failure</th>
<th>NCF failure</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Adhesive failure" /></td>
<td><img src="image2" alt="NCF failure" /></td>
</tr>
<tr>
<td>Displacement: 0.875 mm</td>
<td>Displacement: 1.2 mm</td>
</tr>
<tr>
<td>Displacement: 1.4 mm</td>
<td>Displacement: 1.4 mm</td>
</tr>
</tbody>
</table>

Figure 72: Failure initiation and propagation of the initial geometry of the NCF fabricated H-joint
## Optimization Results

<table>
<thead>
<tr>
<th>Colour</th>
<th>Adhesive</th>
<th>Composite material</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unfailed</td>
<td>Unfailed</td>
<td>Unfailed</td>
</tr>
<tr>
<td>Failure due</td>
<td>Matrix</td>
<td>Fiber</td>
</tr>
<tr>
<td>to shear</td>
<td>stresses</td>
<td>normal</td>
</tr>
<tr>
<td>stresses</td>
<td></td>
<td>stresses</td>
</tr>
</tbody>
</table>

Table 18: Color pattern of failed elements

### 7.1.2. **Optimization**

The tapered length at the H and adhesive thickness, shown in Figure 73 are chosen as DVs for the optimization process. These dimensions are allowed to vary ±50% of their initial value. The other dimensions of the H are excluded in order to comply with manufacturing restrictions related to outer dimensions of the structure and the number of layers in the composite material. To prevent increase in the joint weight, the volume of the H profile is chosen as a SV in the optimization. This practically means that any design set with excessive volume is discarded by the optimization subroutine. From the evaluation of the PDM on the initial geometry it is concluded that the shear stress in the adhesive is responsible for the ultimate failure of the joint. Thus minimization of this stress is set as the OF.
After defining the optimization variables, a simple stress analysis of the initial design is performed and the stress indices at the H and the adhesive are derived using the Maximum Stress criterion (Eq. 8, 9). This step is necessary in order to obtain an initial value for the OF based on which, the candidate designs will be judged.

During the optimization loop 50 design sets are created by the random design generation method. For each design set a stress analysis with a 0.1 mm tensile applied displacement is performed and the calculated stress indices are derived.

7.1.3. **OPTIMAL DESIGN SELECTION**

As discussed also in Chapter 3.2.2 the optimal design must fulfil number of constraints most important of which is the avoidance of failure transition among the different parts. The two different parts in this application are the composite materials that fabricate the joint, and the adhesive. The adhesive shear stresses are proven to cause ultimate failure in the structure as
indicated by the PDM results of the initial geometry. However the critical stress indices of the NCF composite material must be recorded for every design, and be compared to the value of OF to check for failure transition. In Figure 74(a),(b) the maximum stress index for the composite and the adhesive elements for an indicative number of designs are presented.

![Composite and Adhesive Stress Indices](image)

Figure 74: Maximum SI for a) composite material elements and b) adhesive elements for an indicative number of designs

Because no failure transition is observed in any set, the design set that minimizes the value of the OF is chosen as optimum one. The initial and final values of the geometrical optimization parameters are presented in Table 19. In Figure 75, the stress plot of the shear stresses of the adhesive for the initial and the optimal design are depicted. The total reduction in shear adhesive stress in the optimal design 14.2%.
Figure 75: Stress plot of the adhesive elements for a) initial geometry and b) optimal geometry

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Initial value</th>
<th>Final value</th>
</tr>
</thead>
<tbody>
<tr>
<td>DVs</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tapered Length</td>
<td>11 mm</td>
<td>13.1 mm</td>
</tr>
<tr>
<td>Adhesive Thickness</td>
<td>0.195 mm</td>
<td>0.26 mm</td>
</tr>
<tr>
<td>SV</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Element volume</td>
<td>30589 mm³</td>
<td>29389 mm³</td>
</tr>
<tr>
<td>OF</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum normalized adhesive stress</td>
<td>0.819</td>
<td>0.77</td>
</tr>
</tbody>
</table>

Table 19: Initial and final optimization parameters for the NCF fabricated H-joint

## 7.1.4. VERIFICATION OF OPTIMIZATION

The predicted load-displacement curves of the initial and optimum geometries for the NCF material are compared in Figure 76. Optimization is proven to be successful as the failure load and ultimate displacement increase by 6.3% and 9%, respectively. Additionally the weight of the joint is decreased by 4%. The increase in the performance of the joint is attributed to the decrease of shear stress in the adhesive, which leads to a slower debonding progression as can be seen in Figure 77. It is worth noticing that the considerable increase in strength, combined by
the considerable decrease in weight, is achieved by considering rather small variation intervals of the optimization variables.

![Figure 76: Comparison of the force-displacement curves between the initial and optimum geometries for the NCF H-shaped bonded joint](image)

Displacement: 0.875 mm
Figure 77: Comparison of predicted debonding progression between the initial and optimum geometries of the NCF H-shaped bonded joint.
7.2. **JOINING ELEMENT FABRICATED OF 3D FIW COMPOSITE MATERIAL**

7.2.1. **FE MODEL VALIDATION - INITIAL PDM**

As previously stated, due to the lack of experimental data, no comparison with experimental results is conducted in the case of H-joint fabricated of 3D FIW composite material. However, as the FE model of the joint is fully parametric - except for the new material properties - no other changes are needed. Considering this, as well as the fact that the calculated properties of the 3D FIW composite are in very good agreement with the experimental results, the results are considered reliable. The load-displacement of the initial geometry is depicted in Figure 78. The damage progression on the adhesive elements which is responsible for ultimate failure of the structure is shown in Figure 79.

![Figure 78: Load-displacement curve of the 3D FIW H-shaped joining element](image-url)
Debonding is the primary failure mode also in the H-joint with the 3D FIW. However, contrary to the NCF case, no failure is predicted at the element of the H-joint as debonding progression is quicker.
7.2.2. Optimization

The optimization parameters are kept the same as in the NCF case. The optimization process is described in detail in Chapter 6.1.2.

Similarly to the NCF case, the evaluation of the PDM on the initial geometry concluded that the shear stress in the adhesive is responsible for the ultimate failure of the joint. Thus minimization of the particular stress is set as the OF.

7.2.3. Optimal Design Selection

Similarly to the NCF case, the shear adhesive stresses cause ultimate failure in the structure as indicated by the PDM results of the initial geometry. In Figure 80(a),(b) the maximum stress index for the composite and the adhesive elements for an indicative number of designs are presented.

![Graphs showing maximum stress index for composite and adhesive elements](image)

Figure 80: Maximum SI for a) composite material elements and b) adhesive elements for an indicative number of designs.
No failure transition is observed in any set, for the case of the 3D FIW material. Consequently, the design set that minimizes the value of the OF is chosen as optimum one. The initial and final values of the optimization parameters are presented in Table 20. In Figure 81, the stress plot of the shear $\tau_{XZ}$ stresses of the adhesive for the initial and the optimal design are depicted. The achieved reduction in shear adhesive stress is 16.9%.

![Stress plot](image)

**Figure 81:** Stress plot of the a) initial geometry shear $\tau_{XZ}$ stresses and b) optimal geometry shear $\tau_{XZ}$ stresses

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Initial value</th>
<th>Final value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DV</strong>s</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tapered Length</td>
<td>11 mm</td>
<td>16.1 mm</td>
</tr>
<tr>
<td>Adhesive Thickness</td>
<td>0.195 mm</td>
<td>0.258 mm</td>
</tr>
<tr>
<td><strong>SV</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Element volume</td>
<td>30589 mm$^3$</td>
<td>28520 mm$^3$</td>
</tr>
<tr>
<td><strong>OF</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum normalized adhesive stress</td>
<td>0.819</td>
<td>0.66</td>
</tr>
</tbody>
</table>

**Table 20:** Initial and final optimization parameters for the 3D FIW fabricated H-joint
7.2.4. **VERIFICATION OF OPTIMIZATION**

The comparison of the predicted load-displacement curves of the initial and optimum geometries for the 3D FIW material is presented in Figure 82. Like in the NCF case, optimization is proven to be successful. The failure load and ultimate displacement increase by 8.1% and 5.3% respectively. The weight of the joint is reduced by 6.7%. The enhancement in the mechanical properties of the joint is accredited to the slower damage progression in the adhesive as seen in Figure 83.

![Comparison of force-displacement curves](image)

**Figure 82:** Comparison of the force-displacement curves between the initial and optimum geometries for the 3D FIW H-shaped joining element
Optimization Results

7.3. **COMPARISON BETWEEN NCF AND 3D FIW JOINING ELEMENTS**

The fact that both H-shaped bonded joints failed due to debonding reveals the capability of both composite material systems to sustain high axial tensile loads developed at the legs as well as high normal loads developed at the middle of the H. In the case of the 3D FIW bonded joint, no failure is predicted at the joining element due to the large longitudinal tensile and normal strengths of the woven material. By observing the load-displacement curves of both the initial and optimum geometries, shown in Figure 76 and Figure 82, respectively, it is obvious that the NCF H-shaped bonded joint sustains a larger load than the 3D FIW joint which seems contradictory given the previous statement. The larger strength of the NCF joint is attributed to the failure of the weak ±45 NCF layers which release shear stress in the adhesive, thus delaying debonding. In fact, debonding at the 3D FIW bonded joint initiates at the applied displacement of 0.625 mm which is 28% lower than the debonding initiation displacement at the NCF bonded
joint. It is thus concluded that the 3D FIW H is stronger and lighter than the NCF H. However, its superiority is not obvious in the present application because the primary failure mode is not at the composite H-shaped joining element but at the adhesive.
In this thesis, a numerical approach for the SO of composite structural parts with regard to strength while decreasing their initial weight is developed. The proposed numerical approach is a combination of the optimization module of the ANSYS FE code and a PDM module. The methodology is applied in two different applications, both on H-shaped adhesively bonded joints subjected to quasi-static load. In the first application, the H-shaped joining profile is made from NCF composite material while in the second from a novel 3D FIW fabric composite material. The proposed numerical optimization methodology is fully parametric and can be used for any composite structural part under the necessary modifications at its components. In order to apply the PDM module at the macro-scale (global 3D FE model of the H-shaped joining element), a methodology for the numerical characterization of woven composite materials is developed using a mesomechanical approach. Initially the special geometrical characteristics of woven composites are considered to create geometrical models. Consequently, FE models are built and solved using the PDM module. The methodology is parametric and can be applied for any woven composite after making the necessary modifications to the modules.

Using this approach the mechanical response of NCF and 3D FIW composite materials is simulated for all normal and shear loading cases. Especially in the case of 3D FIW this is the first effort to model failure of such composites. Thus, an important contribution of the present work is the derivation of the complete set of the material properties, which, hereafter, can be used in models of structural parts made from the specific 3D FIW composite material. Implementation of the optimization led to joint geometries with higher strengths and smaller weights for both composite material systems. The increase in strength, namely 6.3% for the NCF and 8.1% for the 3D FIW woven joints, is considered significant despite the small variation.
intervals of the optimization variables imposed by manufacturing restrictions. At the same time, the weight is decreased 4% and 6.7% for the NCF and 3D FIW cases respectively.

Finally within the framework of this thesis the advantages of a CDM approach compared to a CZM one, are investigated and assessed. In terms of reliability and applicability it is concluded that CDM is a more advantageous solution as it requires less computational time (50% less) when compared with a CZM and produces highly accurate results with correct meshing and load step values.
Based on the capabilities of the methodologies developed in the present thesis the following subjects are proposed for future study, to either improve the proposed approaches or to implement them in new applications:

- Integration of special optimization algorithms in the optimization routine such as GA algorithms, which can also solve multi objective optimization problems
- Implementation of the optimisation approach to different joining elements such as Pi, L, T, shaped joints.
- Expansion of the current optimization approach for fatigue problems in order to increase the life of a structural part
- Ability to consider composite material characteristics as DVs such as stacking sequence and different types of composite materials
- Implementation of the woven composite modelling approach to other types of composite materials
- Replacement of single mode CDM with a mixed mode one to account for more complex problems
- Consider initial damage of the joint as well as the adhesive via non-destructive testing


References


References


References


Publications

Publications

International Journals


Conferences


   In: 3rd International Conference of Engineering Against Failure (ICEAF III), Kos, Greece, June
   26-28. 2013

4. A.S Koumpias, K.I. Tserpes, Progressive damage modelling of 3D woven composites. In: