

---

# **Static and Dynamic Analysis of an Airplane Wing Representative Structure for Damage Detection**

---

**Beatriz da Silva Henriques**

Dissertation submitted to  
Faculdade de Engenharia da Universidade do Porto  
for the degree of:

Master in Mechanical Engineering

**Advisors:**

Prof. Dr. Pedro Ribeiro  
(Associate Professor)

Prof. Dr. Marcelo Moura  
(Associate Professor)

Departamento de Engenharia Mecânica  
Faculdade de Engenharia da Universidade do Porto

Porto, 2021



# ABSTRACT

---

Over time, the materials and joining technologies used in aeronautical applications evolved and, nowadays, composite materials and bonded joints are very common amongst them. Any existing damage or flaw in these structures jeopardises the system's performance and can even be the reason for its failure and human casualties, especially since there has been a significant augmentation of the operational life of several airplanes. For these reasons, understanding material's failure and improving damage detection technologies has become more important than ever.

In this report, a wing representative structure is analysed. Firstly, a static analysis is performed to identify critical points and components. Next, the critical components, a composite panel and five stiffeners, are isolated and submitted to both progressive damage and implicit dynamic analyses. This aims to identify critical points for damage onset and its propagation and to detect damage existence, respectively, in the composite panel or in the composite panel-stiffener bond. Both analyses are performed in ABAQUS software.

On the progressive damage analysis, a three-point bending situation is recreated and cohesive damage elements are placed in crucial interfaces. A triangular traction-separation law approximates the material's degradation and the stiffener's yielding is accounted for. The results are presented as displacement versus load graphics.

In the implicit dynamic analysis, the structure undergoes harmonic loading and contact restrictions between damaged surfaces are imposed for ensuring reliable results. Time-histories, phase portraits, frequency spectra and Poincaré sections are extracted from the finite element analysis of the structure.

The main conclusions are that the composite panel-long stiffener connection (bonded with Araldite® 420 A/B) is the critical region of the wing representative structure and that the vibration-based health structure monitoring techniques are adequate in the detection of flaws in these structures.

**Keywords:** Wing representative structure, Composite laminate, Bonded joint, Damage, Progressive damage analysis, Cohesive damage model, Plasticity, Nonlinear dynamic analysis, Contact restrictions, Implicit analysis, Finite element analysis.



## RESUMO

---

Ao longo do tempo, os materiais e as tecnologias de ligação utilizados em aplicações aeronáuticas evoluíram e, atualmente, os materiais compósitos e as ligações adesivas são muito comuns nestas aplicações. Qualquer defeito ou dano existente nestas estruturas pode comprometer o seu desempenho e pode mesmo ser o motivo para o seu colapso e para a morte de várias pessoas. A possibilidade de colapso destas estruturas é agravada pelo aumento significativo da vida operacional dos aviões que se tem observado. Assim, perceber a rotura destes materiais e melhorar as tecnologias de deteção de dano é cada vez mais importantes.

Neste relatório, uma estrutura representativa de uma asa de avião é analisada. Primeiro, uma análise estática é desenvolvida com o objetivo de identificar pontos e componentes críticos. De seguida, estes mesmos componentes, um painel laminado e cinco reforços, são isolados e submetidos a uma análise de dano progressivo e a uma análise dinâmica implícita para identificar os locais mais prováveis para a iniciação e propagação de dano e para a deteção de dano, respetivamente. Ambas são realizadas com recurso ao *software* ABAQUS.

Na análise de dano progressivo, é recriada uma situação de flexão em três pontos e os elementos coesivos são colocados nas interfaces consideradas críticas. A lei de tração-separação utilizada para simular a degradação material é triangular e a plastificação dos reforços é também considerada. Os resultados são apresentados sob a forma de gráficos força-deslocamento.

Na análise dinâmica implícita, a estrutura é sujeita a um carregamento harmónico e restrições de contacto são impostas entre as superfícies danificadas para garantir resultados mais fiáveis. São exploradas respostas em função do tempo, trajetórias no plano de fase, secções de Poincaré e espectros de frequência a partir dos resultados da análise de elementos finitos da estrutura.

As principais conclusões são que a interface painel compósito-reforço (unidos através do adesivo Araldite® 420 A/B) é a região crítica da estrutura representativa de uma asa de avião e que os métodos de monitorização da integridade da estrutura baseados em vibrações são adequados para a deteção de defeitos neste tipo de estruturas.

**Palavras-chave:** Estrutura representativa de uma asa de avião, Compósito laminado, Ligação adesiva, Dano, Análise de dano progressivo, Modelo de dano coesivo, Plasticidade, Análise dinâmica não linear, Restrições de contacto, Análise implícita, Análise de elementos finitos



To my parents and boyfriend.



# ACKNOWLEDGMENTS

---

This thesis was developed within the COST action CA18203, “Optimising Design for Inspection”, supported by the European Union Framework Programme Horizon 2020.

In addition, the present report was only possible thanks to the effort of many others, to whom I am very grateful:

- To Prof. Pedro Ribeiro for making me his first choice for this project, for his shared knowledge, encouragement, and availability;
- To Prof. Marcelo Moura for his promptitude to assist me and his incentive to always reach further in my work;
- To the soon to be Dr. Raul Moreira for his valuable advice about the ABAQUS software and his readiness to help;
- To António Carneiro for his help with the mathematical model and his kind words of encouragement;
- To all groups involved in the COST action that contributed to the development of the present work;
- To my friends at FEUP, especially Beatriz Simões and Mariana Carvalho, for all breaks that eased my days and gave me a fresh perspective;
- To my boyfriend, Rui, and my dear family, especially my parents Fátima and Manuel, for their unconditional love and support.



# CONTENTS

---

|                                                              |    |
|--------------------------------------------------------------|----|
| CHAPTER 1. INTRODUCTION 1                                    |    |
| 1.1. BACKGROUND AND MOTIVATION.....                          | 1  |
| 1.2. OPTIMISING DESIGN FOR INSPECTION .....                  | 1  |
| 1.3. OBJETIVES .....                                         | 3  |
| 1.4. LAYOUT OF THE THESIS .....                              | 3  |
| CHAPTER 2. STATE OF THE ART.....                             | 5  |
| 2.1. INTRODUCTION.....                                       | 5  |
| 2.2. PROGRESSIVE DAMAGE MODELLING.....                       | 5  |
| 2.2.1. CLASSICAL PREDICTION METHODS.....                     | 5  |
| 2.2.2. CURRENT PREDICTION METHODS.....                       | 7  |
| 2.3. VIBRATION-BASED STRUCTURAL HEALTH MONITORING.....       | 8  |
| 2.3.1. MODAL-BASED METHODS .....                             | 9  |
| 2.3.2. RESPONSE-BASED METHODS.....                           | 10 |
| 2.3.3. MODEL-BASED METHODS.....                              | 12 |
| 2.4. FINAL REMARKS.....                                      | 14 |
| CHAPTER 3. MATHEMATICAL MODEL .....                          | 15 |
| 3.1. INTRODUCTION.....                                       | 15 |
| 3.2. CONTINUUM MECHANICS .....                               | 15 |
| 3.2.1. KINEMATICS.....                                       | 15 |
| 3.2.2. STRAIN MEASURES.....                                  | 17 |
| 3.2.3. STRESS MEASURES.....                                  | 19 |
| 3.2.4. GOVERNING EQUATIONS.....                              | 20 |
| 3.2.5. CONSTITUTIVE LAWS .....                               | 20 |
| 3.3. CONTACT MECHANICS .....                                 | 28 |
| 3.4. WEAK FORMULATION.....                                   | 30 |
| 3.5. FINITE ELEMENT METHOD.....                              | 32 |
| 3.5.1. SPACE DISCRETISATION .....                            | 32 |
| 3.5.2. TIME DISCRETISATION .....                             | 34 |
| 3.6. FINAL REMARKS.....                                      | 35 |
| CHAPTER 4. WING REPRESENTATIVE STRUCTURE'S DESCRIPTION ..... | 37 |
| 4.1. INTRODUCTION.....                                       | 37 |
| 4.2. WING REPRESENTATIVE STRUCTURE'S GEOMETRY .....          | 37 |

|                                                             |                                                                                                     |    |
|-------------------------------------------------------------|-----------------------------------------------------------------------------------------------------|----|
| 4.3.                                                        | ELASTIC MATERIALS.....                                                                              | 39 |
| 4.3.1.                                                      | FIBRE AND MATRIX CHARACTERISATION.....                                                              | 39 |
| 4.3.2.                                                      | USN150B'S PROPERTIES.....                                                                           | 41 |
| 4.4.                                                        | ELASTOPLASTIC MATERIALS.....                                                                        | 42 |
| 4.5.                                                        | DAMAGED MATERIALS.....                                                                              | 43 |
| 4.6.                                                        | FINAL REMARKS.....                                                                                  | 44 |
| CHAPTER 5. WING REPRESENTATIVE STRUCTURE'S ANALYSIS .....   |                                                                                                     | 45 |
| 5.1.                                                        | INTRODUCTION.....                                                                                   | 45 |
| 5.2.                                                        | STATIC ANALYSIS .....                                                                               | 45 |
| 5.2.1.                                                      | CONVERGENCE STUDY OF THE STATIC ANALYSIS ON THE WING REPRESENTATIVE STRUCTURE .....                 | 46 |
| 5.2.2.                                                      | NUMERICAL RESULTS OF THE STATIC ANALYSIS ON THE WING REPRESENTATIVE STRUCTURE .....                 | 51 |
| 5.3.                                                        | MODAL ANALYSIS .....                                                                                | 56 |
| 5.3.1.                                                      | CONVERGENCE STUDY OF THE MODAL ANALYSIS ON THE WING REPRESENTATIVE STRUCTURE .....                  | 56 |
| 5.3.2.                                                      | NUMERICAL RESULTS OF THE MODAL ANALYSIS ON THE WING REPRESENTATIVE STRUCTURE .....                  | 57 |
| 5.4.                                                        | FINAL REMARKS.....                                                                                  | 59 |
| CHAPTER 6. SUBSTRUCTURE'S PROGRESSIVE DAMAGE ANALYSIS ..... |                                                                                                     | 61 |
| 6.1.                                                        | INTRODUCTION.....                                                                                   | 61 |
| 6.2.                                                        | PROGRESSIVE DAMAGE ANALYSIS.....                                                                    | 61 |
| 6.2.1.                                                      | CONVERGENCE STUDY OF THE PROGRESSIVE DAMAGE ANALYSIS ON THE SUBSTRUCTURE.....                       | 61 |
| 6.2.2.                                                      | NUMERICAL RESULTS OF THE PROGRESSIVE DAMAGE ANALYSIS ON THE SUBSTRUCTURE.....                       | 66 |
| 6.2.3.                                                      | NUMERICAL RESULTS OF THE PROGRESSIVE DAMAGE ANALYSIS ON THE SUBSTRUCTURE WITH AN INITIAL FLAW ..... | 71 |
| 6.3.                                                        | FINAL REMARKS.....                                                                                  | 75 |
| CHAPTER 7. SUBSTRUCTURE'S DYNAMIC ANALYSIS .....            |                                                                                                     | 77 |
| 7.1.                                                        | INTRODUCTION.....                                                                                   | 77 |
| 7.2.                                                        | DAMPING PARAMETERS.....                                                                             | 77 |
| 7.3.                                                        | MODAL ANALYSIS .....                                                                                | 78 |
| 7.4.                                                        | IMPLICIT DYNAMIC ANALYSIS.....                                                                      | 82 |
| 7.4.1.                                                      | CONVERGENCE STUDY OF THE DYNAMIC ANALYSIS ON THE SUBSTRUCTURE.....                                  | 82 |
| 7.4.2.                                                      | NUMERICAL RESULTS OF THE DYNAMIC ANALYSIS ON THE SUBSTRUCTURE – INITIAL DAMAGE SIZE INFLUENCE ..... | 84 |

|                                                                                                             |     |
|-------------------------------------------------------------------------------------------------------------|-----|
| 7.4.3. NUMERICAL RESULTS OF THE DYNAMIC ANALYSIS ON THE SUBSTRUCTURE – MEASUREMENT LOCATION INFLUENCE ..... | 91  |
| 7.4.4. NUMERICAL RESULTS OF THE DYNAMIC ANALYSIS ON THE SUBSTRUCTURE – EXCITATION FREQUENCY INFLUENCE.....  | 96  |
| 7.5. FINAL REMARKS.....                                                                                     | 99  |
| CHAPTER 8. CONCLUSION .....                                                                                 | 101 |
| 8.1. CONCLUSIONS.....                                                                                       | 101 |
| 8.2. FUTURE WORK .....                                                                                      | 102 |
| APPENDIX A CALCULUS OF THE LAMINATE’S ENGINEERING CONSTANTS WITH THE CLASSICAL LAMINATED PLATE THEORY ..... | 111 |



# LIST OF FIGURES

---

|                                                                                                                                                           |    |
|-----------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| <b>Figure 1.1</b> – Wing representative structure. Drawing by School of Engineering, Cardiff University.                                                  | 2  |
| <b>Figure 3.1</b> – Motion of a continuum body $\mathcal{B}$ from the initial to the current configurations. Adapted from Reddy (2013).                   | 16 |
| <b>Figure 3.2</b> – Stress vectors in the initial and current configurations. Adapted from Bonet and Wood (2008).                                         | 19 |
| <b>Figure 3.3</b> – Multiplicative decomposition in the small neighbourhood of a particle. Adapted from Korelc and Wriggers (2016).                       | 23 |
| <b>Figure 3.4</b> – Initial and current elastic domains and yield surfaces. Adapted from Oliver and Saracibar (2017).                                     | 24 |
| <b>Figure 3.5</b> – Triangular pure and mixed-mode cohesive damage model. Adapted from Kashtalyan (2010).                                                 | 26 |
| <b>Figure 3.6</b> – Failure modes. Adapted from Kashtalyan (2010).                                                                                        | 27 |
| <b>Figure 3.7</b> – Contact between two bodies and respective nomenclature at the reference and deformed configurations. Adapted from Wriggers (2006).    | 28 |
| <b>Figure 3.8</b> – Graphical representation of the KKT contact conditions. Adapted from Laursen (2003).                                                  | 30 |
| <b>Figure 4.1</b> – Wing representative structure and its components. Drawing by School of Engineering, Cardiff University.                               | 37 |
| <b>Figure 4.2</b> – Test structure and its components. Drawing by School of Engineering, Cardiff University.                                              | 38 |
| <b>Figure 4.3</b> – Stress-strain approximation for aluminium 6063AT6.                                                                                    | 43 |
| <b>Figure 5.1</b> – Boundary conditions in the static convergence study of the wing representative structure.                                             | 46 |
| <b>Figure 5.2</b> – Imposed displacements in the static convergence study of the wing representative structure.                                           | 46 |
| <b>Figure 5.3</b> – Schematic of the FE used: a) C3D8I and SC8R; b) C3D10.                                                                                | 48 |
| <b>Figure 5.4</b> – Load-displacement graphic, for different average element sizes, of the static convergence study on the wing representative structure. | 48 |
| <b>Figure 5.5</b> – FE model of the wing representative structure and used elements in the static convergence study.                                      | 50 |

|                                                                                                                                                                                                                                                                  |    |
|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| <b>Figure 5.6</b> – Displacement results of the static convergence study on the wing representative structure, for imposed displacements of: a) 5 mm; b) 7 mm. Magnified 3x. ....                                                                                | 50 |
| <b>Figure 5.7</b> – FE model of the wing representative structure and used elements in the static analysis. ....                                                                                                                                                 | 52 |
| <b>Figure 5.8</b> – Load-displacement graphic of the static analysis on the wing representative structure. ....                                                                                                                                                  | 52 |
| <b>Figure 5.9</b> – Stress results of the static analysis, for an imposed displacement of 6.12 mm, on the: a) wing representative structure; b) testing structure. ....                                                                                          | 53 |
| <b>Figure 5.10</b> – Plastic strain results of the static analysis, for an impose displacement of 6.12 mm, on the aluminium stiffeners.....                                                                                                                      | 54 |
| <b>Figure 5.11</b> – Cohesive elements location in the stiffeners-panels interface mimicking the adhesive’s behaviour. ....                                                                                                                                      | 54 |
| <b>Figure 5.12</b> – Detail of the displacement results of the static analysis, for an imposed displacement of 6.12 mm, on the cohesive elements. ....                                                                                                           | 55 |
| <b>Figure 5.13</b> – Original wing representative structure to the left and the new chosen substructure on the right. ....                                                                                                                                       | 55 |
| <b>Figure 5.14</b> – 7 <sup>th</sup> mode shape of the wing representative structure for $\omega = 145.57 \text{ rad.s} - 1$ . ....                                                                                                                              | 57 |
| <b>Figure 5.15</b> – 8 <sup>th</sup> mode shape of the wing representative structure for $\omega = 213.89 \text{ rad.s} - 1$ . ....                                                                                                                              | 58 |
| <b>Figure 5.16</b> – 9 <sup>th</sup> mode shape of the wing representative structure for $\omega = 248.58 \text{ rad.s} - 1$ . ....                                                                                                                              | 58 |
| <b>Figure 5.17</b> – 10 <sup>th</sup> mode shape of the wing representative structure for $\omega = 357.68 \text{ rad.s} - 1$ . ..                                                                                                                               | 58 |
| <b>Figure 5.18</b> – 11 <sup>th</sup> mode shape of the wing representative structure for $\omega = 375.82 \text{ rad.s} - 1$ . ..                                                                                                                               | 58 |
| <b>Figure 6.1</b> – Boundary conditions and imposed displacements in the progressive damage convergence study of the substructure. ....                                                                                                                          | 62 |
| <b>Figure 6.2</b> – Load-displacement graphic, for different average element sizes, of the progressive damage convergence study on the substructure. ....                                                                                                        | 63 |
| <b>Figure 6.3</b> – FE model of the substructure and used elements in the progressive damage convergence study. ....                                                                                                                                             | 64 |
| <b>Figure 6.4</b> – Von Mises stress results of the progressive damage convergence study, for an imposed displacement of 50 mm, on the substructure.....                                                                                                         | 64 |
| <b>Figure 6.5</b> – Von Mises stress results of the progressive damage convergence study, for an imposed displacement of 50 mm, through the thickness: a) of the substructure (composite panel + stiffener) in the centre region; b) of the composite panel..... | 65 |
| <b>Figure 6.6</b> – FE model of the substructure and used elements in the progressive damage analysis. ....                                                                                                                                                      | 67 |
| <b>Figure 6.7</b> – Load-displacement graphic of the progressive damage analysis on the substructure. ....                                                                                                                                                       | 67 |
| <b>Figure 6.8</b> – Plastic strain results of the progressive damage analysis, for an imposed displacement of 43.8 mm, on the substructure. ....                                                                                                                 | 68 |

|                                                                                                                                                                                                                                                                                                                  |    |
|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| <b>Figure 6.9</b> – Plastic strain (a) and damage parameter (b) results of the progressive damage analysis, for an imposed displacement of 129.6 mm, on the substructure. ....                                                                                                                                   | 69 |
| <b>Figure 6.10</b> – Plastic strain (a) and damage parameter (b) results of the progressive damage analysis, for an imposed displacement of 132.4 mm, on the substructure. ....                                                                                                                                  | 70 |
| <b>Figure 6.11</b> – FE model of the substructure and used elements in the progressive damage analysis with an initial flaw. ....                                                                                                                                                                                | 71 |
| <b>Figure 6.12</b> – Contact surfaces between the composite panel and the stiffeners in the progressive damage analysis of the substructure with an initial flaw: in pink the slave surface and in red the master one. Even though only one stiffener is represented, the same occurs for all five of them. .... | 72 |
| <b>Figure 6.13</b> – Load-displacement graphic of the progressive damage analysis on the substructure with different initial flaws. ....                                                                                                                                                                         | 72 |
| <b>Figure 6.14</b> – Plastic strain results of the progressive damage analysis on the substructure with different initial flaw sizes: (a) $\gamma = 0 \%$ ; (b) $\gamma = 2 \%$ ; (c) $\gamma = 10 \%$ ; (d) $\gamma = 20 \%$ . ....                                                                             | 73 |
| <b>Figure 6.15</b> – Damage parameter results of the progressive damage analysis on the substructure with different initial flaw sizes: (a) $\gamma = 0 \%$ ; (b) $\gamma = 2 \%$ ; (c) $\gamma = 10 \%$ ; (d) $\gamma = 20 \%$ . ....                                                                           | 74 |
| <b>Figure 7.1</b> – Boundary conditions in the modal analysis of the stiffener. ....                                                                                                                                                                                                                             | 77 |
| <b>Figure 7.2</b> – Tie interaction between the composite panel and the stiffeners in the modal analysis of the substructure. The central damaged region is not tied. ....                                                                                                                                       | 78 |
| <b>Figure 7.3</b> – Boundary conditions in the modal analysis of the substructure. ....                                                                                                                                                                                                                          | 79 |
| <b>Figure 7.4</b> – $i$ th mode shape of the substructure: a) $i = 1$ ; b) $i = 2$ ; c) $i = 3$ ; d) $i = 4$ ; e) $i = 5$ ; f) $i = 6$ . ....                                                                                                                                                                    | 80 |
| <b>Figure 7.5</b> – Harmonic loading applied on the central region in the substructure for the dynamic analysis. ....                                                                                                                                                                                            | 82 |
| <b>Figure 7.6</b> – Time-history of the dynamic converge study on the substructure, for a different number of discretisation points per wave, at $\omega = 500 \text{ rad.s}^{-1}$ . ....                                                                                                                        | 83 |
| <b>Figure 7.7</b> – Phase portrait of the dynamic converge study on the substructure, for a different number of discretisation points per wave, at $\omega = 500 \text{ rad.s}^{-1}$ . ....                                                                                                                      | 84 |
| <b>Figure 7.8</b> – Contact surface between the composite panel and the stiffener in the dynamic analysis on the substructure: in pink the slave surface and in red the master one. ....                                                                                                                         | 85 |
| <b>Figure 7.9</b> – Time-history of dynamic analysis on the substructure at $\omega = 500 \text{ rad.s}^{-1}$ with initial flaws of $\gamma = 0 \%$ and: a) $\gamma = 0.4 \%$ ; b) $\gamma = 2 \%$ ; c) $\gamma = 4 \%$ ; d) $\gamma = 8 \%$ . ....                                                              | 86 |
| <b>Figure 7.10</b> – Phase portraits of the dynamic analysis on the substructure at $\omega = 500 \text{ rad.s}^{-1}$ with initial flaws of $\gamma = 0 \%$ and: a) $\gamma = 0.4 \%$ ; b) $\gamma = 2 \%$ ; c) $\gamma = 4 \%$ ; d) $\gamma = 8 \%$ . ....                                                      | 87 |
| <b>Figure 7.11</b> – Frequency spectrum of the dynamic analysis on the substructure at $\omega = 500 \text{ rad.s}^{-1}$ with initial flaws of $\gamma = 0 \%$ and: a) $\gamma = 0.4 \%$ ; b) $\gamma = 2 \%$ ; c) $\gamma = 4 \%$ ; d) $\gamma = 8 \%$ . ....                                                   | 88 |
| <b>Figure 7.12</b> – Poincaré sections of the dynamic analysis on the substructure at $\omega = 500 \text{ rad.s}^{-1}$ with initial flaws of $\gamma = 0 \%$ and: a) $\gamma = 0.4 \%$ ; b) $\gamma = 2 \%$ ; c) $\gamma = 4 \%$ ; d) $\gamma = 8 \%$ . ....                                                    | 89 |

|                                                                                                                                                                                                                                       |     |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-----|
| <b>Figure 7.13</b> – Time-history of the detached parts on the dynamic analysis at $\omega = 500 \text{ rad.s}^{-1}$ with an initial flaw of $\gamma = 8 \%$ . .....                                                                  | 90  |
| <b>Figure 7.14</b> – Location of the twelve measurement points in the substructure for the dynamic analysis. ....                                                                                                                     | 91  |
| <b>Figure 7.15</b> – Time-history of the dynamic analysis on the substructure with initial flaws of $\gamma = 0 \%$ and $\gamma = 8 \%$ at $\omega = 500 \text{ rad.s}^{-1}$ in point: a) 1; b) 2; c) 3; d) 4; e) 5; f) 6. ....       | 92  |
| <b>Figure 7.16</b> – Time-history of the dynamic analysis on the substructure with initial flaws of $\gamma = 0 \%$ and $\gamma = 8 \%$ at $\omega = 500 \text{ rad.s}^{-1}$ in point: a) 7; b) 8; c) 9; d) 10; e) 11; f) 12. ....    | 93  |
| <b>Figure 7.17</b> – Frequency spectrum of the dynamic analysis on the substructure with initial flaws of $\gamma = 0 \%$ and $\gamma = 8 \%$ at $\omega = 500 \text{ rad.s}^{-1}$ in point: a) 1; b) 4. ....                         | 94  |
| <b>Figure 7.18</b> – Poincaré sections of the dynamic analysis on the substructure with initial flaws of $\gamma = 0 \%$ and $\gamma = 8 \%$ at $\omega = 500 \text{ rad.s}^{-1}$ in point: a) 1; b) 2; c) 3; d) 4; e) 5; f) 6. ....  | 95  |
| <b>Figure 7.19</b> – Time-history (a) and phase portrait (b) of the dynamic convergence study on the substructure, for a different number of discretisation points per wave, at $\omega = 1000 \text{ rad.s}^{-1}$ . ....             | 96  |
| <b>Figure 7.20</b> – Time-history of the dynamic analysis on the substructure with initial flaws of $\gamma = 0 \%$ and $\gamma = 8 \%$ at: a) $\omega = 500 \text{ rad.s}^{-1}$ ; b) $\omega = 1000 \text{ rad.s}^{-1}$ . ....       | 97  |
| <b>Figure 7.21</b> – Time-history of the detached parts on dynamic analysis at $\omega = 1000 \text{ rad.s}^{-1}$ for an initial flaw of $\gamma = 8 \%$ . ....                                                                       | 97  |
| <b>Figure 7.22</b> – Frequency spectrum of the dynamic analysis on the substructure with initial flaws of $\gamma = 0 \%$ and $\gamma = 8 \%$ at: a) $\omega = 500 \text{ rad.s}^{-1}$ ; b) $\omega = 1000 \text{ rad.s}^{-1}$ . .... | 98  |
| <b>Figure 7.23</b> – Phase portrait of the dynamic analysis on the substructure with initial flaws of $\gamma = 0 \%$ and $\gamma = 8 \%$ at: a) $\omega = 500 \text{ rad.s}^{-1}$ ; b) $\omega = 1000 \text{ rad.s}^{-1}$ . ....     | 98  |
| <b>Figure A.1</b> – Coordinate systems used. ....                                                                                                                                                                                     | 111 |

## LIST OF TABLES

---

|                                                                                                                                                                                                        |    |
|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| <b>Table 4.1</b> – List of the structure’s constituents and their materials. ....                                                                                                                      | 38 |
| <b>Table 4.2</b> – Elastic properties of aluminium and steel alloys and of Araldite® 420 A/B. ....                                                                                                     | 39 |
| <b>Table 4.3</b> – Elastic properties of TR50S15L fibres and SKYFLEX K51 resin. ....                                                                                                                   | 39 |
| <b>Table 4.4</b> – USN150A’s composition and elastic properties. ....                                                                                                                                  | 40 |
| <b>Table 4.5</b> – USN150B’s composition. ....                                                                                                                                                         | 41 |
| <b>Table 4.6</b> – USN150B’s elastic properties. ....                                                                                                                                                  | 42 |
| <b>Table 4.7</b> – Summary of the damage parameters chosen for USN150B and Araldite® 420 A/B. ....                                                                                                     | 44 |
| <b>Table 5.1</b> – Number of elements and DOF used, and time spent per increment for different average element sizes in the static analysis of the wing representative structure. ....                 | 49 |
| <b>Table 5.2</b> – Initial stiffness and buckling point coordinates for different average element sizes in the static analysis of the wing representative structure. ....                              | 49 |
| <b>Table 5.3</b> – Maximum Hashin’s damage parameters in the static analysis of the wing representative structure. ....                                                                                | 54 |
| <b>Table 5.4</b> – Number of elements and DOF used, and time spent for different average element sizes in the modal analysis of the wing representative structure. ....                                | 56 |
| <b>Table 5.5</b> – First five natural frequencies (rather than 0) obtained for different average element sizes in the modal analysis of the wing representative structure. ....                        | 57 |
| <b>Table 6.1</b> – Number of elements and DOF used, time spent, and maximum load for different average element sizes in the progressive damage analysis of the substructure. ....                      | 63 |
| <b>Table 6.2</b> – Maximum load verified for different initial flaw sizes in the progressive damage analysis of the substructure. ....                                                                 | 75 |
| <b>Table 7.1</b> – Proportional damping parameters. ....                                                                                                                                               | 78 |
| <b>Table 7.2</b> – First six natural frequencies obtained for different initial flaw sizes in the modal analysis of the substructure. ....                                                             | 79 |
| <b>Table 7.3</b> – Parameters for the HHT integrator used in the dynamic analysis of the substructure. ....                                                                                            | 82 |
| <b>Table 7.4</b> – Time increments used, and time spent for different number of discretisation points per wave at $\omega = 500 \text{ rad.s}^{-1}$ in the dynamic analysis of the substructure. ....  | 83 |
| <b>Table 7.5</b> – Time increments used, and time spent for different number of discretisation points per wave at $\omega = 1000 \text{ rad.s}^{-1}$ in the dynamic analysis of the substructure. .... | 96 |



# NOMENCLATURE / LIST OF SYMBOLS

---

## General Abbreviations

|      |                                                              |
|------|--------------------------------------------------------------|
| FPZ  | Fracture <b>P</b> rocess <b>Z</b> one                        |
| SHM  | Structural <b>H</b> ealth <b>M</b> onitoring                 |
| VSHM | Vibration-based Structural <b>H</b> ealth <b>M</b> onitoring |
| ANN  | <b>A</b> rtificial <b>N</b> eural <b>N</b> etworks           |
| PCA  | <b>P</b> rincipal <b>C</b> omponent <b>A</b> nalysis         |
| GA   | <b>G</b> enetic <b>A</b> lgorithms                           |
| FEM  | <b>F</b> inite <b>E</b> lement <b>M</b> ethod                |
| FE   | <b>F</b> inite <b>E</b> lement                               |
| KKT  | <b>K</b> arush– <b>K</b> uhn– <b>T</b> ucker                 |
| HHT  | <b>H</b> ilber– <b>H</b> ughes– <b>T</b> aylor               |
| PVW  | <b>P</b> rinciple of <b>V</b> irtual <b>W</b> ork            |

## General Notation

|                           |                                        |
|---------------------------|----------------------------------------|
| $a, A$                    | Scalar ( <i>italic</i> )               |
| $\mathbf{a}, \{\bullet\}$ | Vector ( <i>italic</i> + <b>bold</b> ) |
| $\mathbf{A}, [\bullet]$   | Second-order tensor ( <b>bold</b> )    |
| $\mathbb{A}$              | Fourth-order tensor                    |

## Operators

|                                            |                                              |
|--------------------------------------------|----------------------------------------------|
| $\bullet^{-1}$                             | Inverse of a tensor/matrix                   |
| $\dot{\bullet} = d\bullet/dt$              | First total time derivative                  |
| $\ddot{\bullet} = d^2\bullet/dt^2$         | Second total time derivative                 |
| $\partial\bullet/\partial\bullet$          | Partial derivative                           |
| $\nabla_{x^0}(\bullet), \nabla_x(\bullet)$ | Material and spatial gradient operators      |
| $\text{skew}(\bullet)$                     | Skew component of a second-order tensor      |
| $\text{symm}(\bullet)$                     | Symmetric component of a second-order tensor |
| $\mathbf{I}$                               | Second-order identity tensor                 |
| $\det(\bullet)$                            | Determinant                                  |
| $\bullet^{-T}$                             | Inverse of the transpose of a tensor/matrix  |

|                                                    |                                                      |
|----------------------------------------------------|------------------------------------------------------|
| $\bullet^T$                                        | Transpose of a tensor/matrix                         |
| $\Delta(\bullet)$                                  | Increment                                            |
| $\text{div}_{x^0}(\bullet), \text{div}_x(\bullet)$ | Material and spatial divergence operators            |
| $(\bullet) : (\bullet)$                            | Double contraction / dot product between two tensors |
| $\text{sym.}$                                      | Symmetric entity                                     |
| $\ \bullet\ $                                      | Euclidean vector norm                                |
| $H$                                                | Sobolev function space                               |
| $\delta(\bullet)$                                  | First variation/Virtual quantity                     |
| $(\bullet) \cdot (\bullet)$                        | Single contraction / dot product between two tensors |

### Subscripts and Superscripts

|                   |                                   |
|-------------------|-----------------------------------|
| $(\bullet)^0$     | In the initial configuration      |
| $(\bullet)^e$     | Elastic                           |
| $(\bullet)^p$     | Plastic                           |
| $(\bullet)_{ref}$ | Reference                         |
| $(\bullet)_y$     | Yield                             |
| $(\bullet)_d$     | Damage                            |
| $(\bullet)_c$     | Contact, composite or compressive |
| $(\bullet)_C$     | Critical                          |
| $(\bullet)_{kin}$ | Kinetic                           |
| $(\bullet)_{int}$ | Internal                          |
| $(\bullet)_{ext}$ | External                          |
| $(\bullet)_e$     | Element discrete quantity         |
| $(\bullet)_h$     | Finite element approximation      |
| $(\bullet)_m$     | Master                            |
| $(\bullet)_s$     | Slave                             |
| $(\bullet)_f$     | Fibre                             |
| $(\bullet)_R$     | Resin or matrix                   |
| $(\bullet)_{A,B}$ | USN150A and USN150B               |
| $(\bullet)_T$     | Total                             |
| $(\bullet)_t$     | Traction                          |
| $(\bullet)_k$     | Each layer $k$                    |
| $(\bullet)_n$     | Index associated with time $t_n$  |
| $(\bullet)_j$     | Nonlinear iteration index         |

## Accents

|                   |                        |
|-------------------|------------------------|
| ( $\hat{\circ}$ ) | Projected              |
| ( $\circ$ )       | Unloaded configuration |
| ( $\bullet$ )     | Prescribed             |

## General Physical Symbols

|        |         |
|--------|---------|
| $\rho$ | Density |
| $m$    | Mass    |
| $t$    | Time    |

## ABAQUS' Outputs

|          |                                                                  |
|----------|------------------------------------------------------------------|
| U        | ABAQUS' Displacement Outputs                                     |
| S        | ABAQUS' Stress Outputs                                           |
| LE       | ABAQUS' Logarithmic Strain Outputs                               |
| RF       | ABAQUS' Reaction Force Outputs                                   |
| PE       | ABAQUS' Plastic Strain Outputs                                   |
| HSNFTCRT | ABAQUS' Hashin's Fibre Tensile Initiation Criterion Outputs      |
| HSNFCCRT | ABAQUS' Hashin's Fibre Compressive Initiation Criterion Outputs  |
| HSNMTCRT | ABAQUS' Hashin's Matrix Tensile Initiation Criterion Outputs     |
| HSNMCCRT | ABAQUS' Hashin's Matrix Compressive Initiation Criterion Outputs |
| SDEG     | ABAQUS' Scalar Stiffness Degradation                             |

## CHAPTER 2.2.1 – CLASSICAL PREDICTION METHODS

|          |                                                       |
|----------|-------------------------------------------------------|
| $K$      | Stress intensity factor                               |
| $K_C$    | Fracture toughness                                    |
| $G_i$    | Strain energy release rate for pure mode $i$          |
| $G_{iC}$ | Critical strain energy release rate for pure mode $i$ |

## CHAPTER 3.2.1 - KINEMATICS

|                            |                                                               |
|----------------------------|---------------------------------------------------------------|
| $\mathbb{R}$               | Real set                                                      |
| $\Omega^0, \Omega$         | Domain of a body in the initial and current configurations    |
| $P, p$                     | General point in the initial and current configurations       |
| $\varphi$                  | Deformation function                                          |
| $\mathbf{x}^0, \mathbf{x}$ | Position vector in the initial and current configurations     |
| $\mathbf{u}^0, \mathbf{u}$ | Displacement vector in the initial and current configurations |

|                                       |                                  |
|---------------------------------------|----------------------------------|
| $\dot{\mathbf{x}}, \ddot{\mathbf{x}}$ | Velocity and acceleration vector |
| $\mathbf{l}$                          | Spatial velocity gradient        |
| $\mathbf{d}$                          | Rate of deformation tensor       |
| $\mathbf{w}$                          | Spin tensor                      |

### CHAPTER 3.2.2 – STRAIN MEASURES

|                              |                                                         |
|------------------------------|---------------------------------------------------------|
| $\mathbf{F}$                 | Deformation gradient                                    |
| $d\mathbf{x}^0, d\mathbf{x}$ | Line element in the initial and current configurations  |
| $J$                          | Jacobian determinant, determinant of $\mathbf{F}$       |
| $d\mathbf{a}^0, d\mathbf{a}$ | Area element in the initial and current configurations  |
| $dv^0, dv$                   | Volume element in the initial and current configuration |
| $\mathbf{R}$                 | Rotation tensor                                         |
| $\mathbf{U}, \mathbf{V}$     | Right and left stretch tensor                           |
| $\mathbf{C}, \mathbf{b}$     | Right and left Cauchy-Green deformation tensor          |
| $\mathbf{E}$                 | Green-Lagrange strain tensor                            |
| $\mathbf{e}$                 | Euler-Almansi strain tensor                             |
| $\boldsymbol{\varepsilon}$   | Small strain tensor                                     |
| $\dot{\mathbf{E}}$           | Material strain rate tensor                             |
| $\dot{\mathbf{F}}$           | Time derivative of the deformation gradient             |

### CHAPTER 3.2.3 – STRESS MEASURES

|                            |                                                                                |
|----------------------------|--------------------------------------------------------------------------------|
| $\mathbf{n}^0, \mathbf{n}$ | Outward normal vector in the initial and current configurations                |
| $\mathbf{f}$               | Resultant force                                                                |
| $\mathbf{t}^0, \mathbf{t}$ | Traction vector relative to the area in the initial and current configurations |
| $\boldsymbol{\sigma}$      | Cauchy stress tensor                                                           |
| $\boldsymbol{\tau}$        | Kirchhoff stress tensor                                                        |
| $\mathbf{P}$               | First Piola-Kirchhoff stress tensor                                            |
| $\mathbf{S}$               | Second Piola-Kirchhoff stress tensor                                           |

### CHAPTER 3.2.5 – CONSTITUTIVE LAWS

#### Elastic Materials

|              |                                                                              |
|--------------|------------------------------------------------------------------------------|
| $\Psi$       | Strain energy function                                                       |
| $\mathbb{K}$ | Fourth-order tensor containing the elastic stiffness coefficients $K_{ijkl}$ |
| $E$          | Young's modulus                                                              |
| $\nu$        | Poisson's ratio                                                              |
| $G$          | Shear modulus                                                                |

### Elastoplastic Materials

|                                                       |                                                                    |
|-------------------------------------------------------|--------------------------------------------------------------------|
| $\mathbf{F}^e, \mathbf{F}^p$                          | Elastic and plastic components of the deformation gradient         |
| $\tilde{\mathbf{C}}^e, \mathbf{C}^p$                  | Elastic and plastic components of the right Cauchy-Green tensor    |
| $f$                                                   | Yield function                                                     |
| $\sigma_y, \sigma_{ref}$                              | Initial and current yield stresses                                 |
| $\alpha_p$                                            | Hardening variable                                                 |
| $\mathbf{s}$                                          | Deviatoric component of the Kirchhoff stress                       |
| $p$                                                   | Hydrostatic pressure                                               |
| $\mathbb{E}^0, \mathbb{E}^{\alpha_p}$                 | Initial and current elastic domain                                 |
| $\partial\mathbb{E}^0, \partial\mathbb{E}^{\alpha_p}$ | Initial and current elastic boundary                               |
| $H$                                                   | Material-hardening parameter                                       |
| $\mathbf{d}^e, \mathbf{d}^p$                          | Elastic and plastic components of the rate of deformation gradient |
| $\lambda$                                             | Plastic multiplier                                                 |
| $\dot{\alpha}_p$                                      | Hardening rate                                                     |

### Damaged Materials

|                              |                                                                                |
|------------------------------|--------------------------------------------------------------------------------|
| $\Gamma_d^0, \Gamma_d$       | Boundary potential damaged partition in the initial and current configurations |
| $\mathbf{t}_d$               | Cohesive traction                                                              |
| $\delta_r$                   | Relative displacement or separation                                            |
| $\mathbb{K}_d$               | Stiffness fourth order tensor containing the stiffness coefficients            |
| $\delta_{o,i}, \delta_{u,i}$ | Onset and ultimate relative displacement in pure mode $i$                      |
| $\mathbf{D}$                 | Diagonal matrix containing the damage parameters $d_i$                         |
| $d_i$                        | Damage parameter for pure mode $i$                                             |
| $\sigma_{u,i}$               | Mode $i$ strength                                                              |
| $\varphi$                    | Exponent of the power law                                                      |

## CHAPTER 3.3 – CONTACT MECHANICS

|                                    |                                                                                |
|------------------------------------|--------------------------------------------------------------------------------|
| $\partial\Omega^0, \partial\Omega$ | Boundary of a body in the initial and current configurations                   |
| $\Gamma_u^0, \Gamma_u$             | Dirichlet partition of the boundary in the initial and current configurations  |
| $\Gamma_\sigma^0, \Gamma_\sigma$   | Neumann partition of the boundary in the initial and current configurations    |
| $\Gamma_c^0, \Gamma_c$             | Boundary potential contact partition in the initial and current configurations |
| $\Gamma_a^0, \Gamma_a$             | Boundary active contact partition in the initial and current configurations    |
| $g_n$                              | Gap function                                                                   |
| $\mathbf{n}_c$                     | Slave surface contact normal vector                                            |
| $\mathbf{t}_c$                     | Contact traction vector in the current configuration                           |
| $p_n$                              | Normal contact pressure in the current configuration                           |
| $\dot{g}_n$                        | Material time derivative of the gap function                                   |

## CHAPTER 3.4 – WEAK FORMULATION

|                            |                                                                   |
|----------------------------|-------------------------------------------------------------------|
| $\mathcal{U}$              | Solution space                                                    |
| $\mathcal{V}$              | Weighting space                                                   |
| $\delta\Pi$                | Virtual work                                                      |
| $\delta\mathbf{u}$         | Virtual displacement field                                        |
| $\mathbf{t}^0, \mathbf{t}$ | Surface traction vector in the initial and current configurations |
| $\mathbf{b}_0, \mathbf{b}$ | Body force in the initial and current configurations              |
| $\bar{\mathbf{u}}_i^0$     | Initial velocity field                                            |
| $\bar{\mathbf{u}}_i^0$     | Initial displacement field                                        |
| $\lambda$                  | Lagrange multiplier vector                                        |
| $\mathcal{M}$              | Lagrange multiplier solution space                                |
| $\mathcal{W}$              | Trace space of the weighting space                                |
| $\delta\lambda$            | Trial Lagrange multiplier vector                                  |

## CHAPTER 3.5.1 – SPACE DISCRETISATION

|                                                   |                                                                      |
|---------------------------------------------------|----------------------------------------------------------------------|
| $n_e$                                             | Number of elements                                                   |
| $N_k$                                             | Local interpolation function, shape function of node $k$             |
| $\mathbf{d}_k$                                    | Nodal displacement vector of node $k$                                |
| $n_{nod}$                                         | Number of nodes                                                      |
| $\mathbf{M}, \mathbf{C}$                          | Global mass and damping matrixes                                     |
| $\mathbf{F}$                                      | Global vector of nodal forces                                        |
| $\mathbf{d}, \dot{\mathbf{d}}, \ddot{\mathbf{d}}$ | Global vectors of nodal displacements, velocities, and accelerations |
| $\mathbf{M}_e, \mathbf{C}_e$                      | Elemental mass and damping matrixes                                  |
| $\mathbf{K}_e^{init}$                             | Elemental initial tangent stiffness matrix                           |
| $\alpha, \beta$                                   | Mass and stiffness matrix coefficients in Rayleigh damping model     |
| $\mathbf{d}_0, \dot{\mathbf{d}}_0$                | Initial nodal vectors of displacements and velocities                |

## CHAPTER 3.5.2 – TIME DISCRETISATION

|                                                         |                                                                  |
|---------------------------------------------------------|------------------------------------------------------------------|
| $\Delta t$                                              | Time step                                                        |
| $\alpha_{HHT}$                                          | Parameter of HHT method                                          |
| $\mathbf{d}_n, \dot{\mathbf{d}}_n, \ddot{\mathbf{d}}_n$ | Nodal displacements, velocities, and accelerations at time $t_n$ |
| $\beta_N, \gamma_N$                                     | Parameters of Newmark's method                                   |

#### **CHAPTER 4.3.1 – FIBRE AND MATRIX CHARACTERISATION**

$RC, FC$  Resin and fibre contents

$V_f$  Fibre volume fraction

#### **CHAPTER 4.3.2 – USN150B’S PROPERTIES**

$X, Y, Z$  Normal strengths in direction 1, 2 and 3

$S$  Shear strength

$t_{ply}$  Ply thickness

#### **CHAPTER 4.4 – ELASTOPLASTIC MATERIALS**

$\sigma_R$  Ultimate stress

$\varepsilon_f$  Elongation at break

#### **CHAPTER 5.2 – STATIC ANALYSIS**

$\mathbb{A}, \mathbb{B}, \mathbb{D}$  Laminate’s stiffness matrices

$n_{dof}$  Number of degrees of freedom (DOF)

$E_{in}$  Initial stiffness of the structure

#### **CHAPTER 5.3 – MODAL ANALYSIS**

$\omega_i$   $i^{\text{th}}$  natural frequency

#### **CHAPTER 6.2 – PROGRESSIVE DAMAGE ANALYSIS**

$\mu$  Viscosity parameter

$l_{in}$  Initial flaw length

$\gamma$  Damaged area parameter

#### **CHAPTER 7.2 – DAMPING PARAMETERS**

$\xi$  Viscous damping ratio



# CHAPTER 1. INTRODUCTION

---

## 1.1. BACKGROUND AND MOTIVATION

Over time, the materials used in mechanical constructions evolved from steel to aluminium to composites and hybrid materials, especially in aeronautical and aerospace applications. This was motivated by the need to develop lighter structures with high specific mechanical properties (Kashtalyan, 2010). Also, in a structure's assembly, joining technologies such as bolted joints or welding evolved to bonded or hybrid joints, because these have less sources of stress concentrations, distribute loads more uniformly and have better fatigue properties (da Silva and Öchsner, 2008).

Flaws are very common in composite materials and in bonded joints. In composites, they can either be produced during the manufacturing process or their service life. Examples of defects are porosity, caused by non-optimal cure parameters, or delaminations, most of the time caused by impact situations (Smith, 2009g). In bonded joints, flaws are usually formed during the manufacturing process: inadequate joint preparation, air bubbles, foreign bodies or degradation due to environmental conditions (Ribeiro et al., 2016).

Such imperfections not only compromise these materials' integrity, but also the integrity of the entire structure and ultimately being the reason for its failure. When these substructures are components from aeronautical and aerospace applications, the result may be human casualties. Consequently, there is the need for understanding and predicting these materials' failure. Since the operational life of these vehicles has been increased, controlling damage propagation has never been more important.

Thus, the main subject of the present work is to perform analyses on a wing representative structure. The structure is composed of aluminium and composite materials and assembled with hybrid joints. The goal is to identify critical points for damage onset and propagation and infer about the use of nonlinear vibration-based structural health monitoring (VSHM) technologies.

## 1.2. OPTIMISING DESIGN FOR INSPECTION

The European Cooperation in Science and Technology (COST) is a funding organisation for the creation of research networks, called COST Actions. These networks encourage the collaboration between scientists across Europe and thereby give impetus to research advancements and innovation. To this matter, COST receives EU funding under the various research and innovation framework programmes, such as Horizon 2020.

The present thesis is developed within the EU COST action CA18203, "Optimising Design for Inspection" that brings together European experts across the key areas of damage detection, energy

harvesting, wireless communication, and optimisation. The goal is to support the development of an integrated framework for optimised self-sensing structures capable of diagnosis and prognosis, together with demonstrators and educational activities, including training programs, which ultimately lead to cleaner and safer skies (European COST, 2019).

This action is divided into five working groups, each one with their own tasks and goals:

- Working Group 1 – Establish design criteria, based on industry requirements for a representative wing structure element and analyse the requirements for integrating structural health monitoring (SHM) systems at the inception of an aerospace design;
- Working Group 2 – Analyse existing strategies including sensor technologies and deliver a strategy for sharing data and signal processing algorithms;
- Working Group 3 – Develop a detailed understanding of current vibration levels and temperature differences and the location or position they would be found on an aircraft and standard testing procedures to allow a comparison across European research groups;
- Working Group 4 – Review aerospace protocols and strategies that will reduce power requirements at a sensor node;
- Working Group 5 – This working group focus on human interface, data interpretation, data presentation, data mining, data efficiency/reduction and hardware integration.

The present work integrates the Working Group 1 (WG1).

The wing representative structure's geometry and materials is proposed by WG1 colleagues from Cardiff School of Engineering, after deliberating with the company Airbus. The result is represented in Figure 1.1.

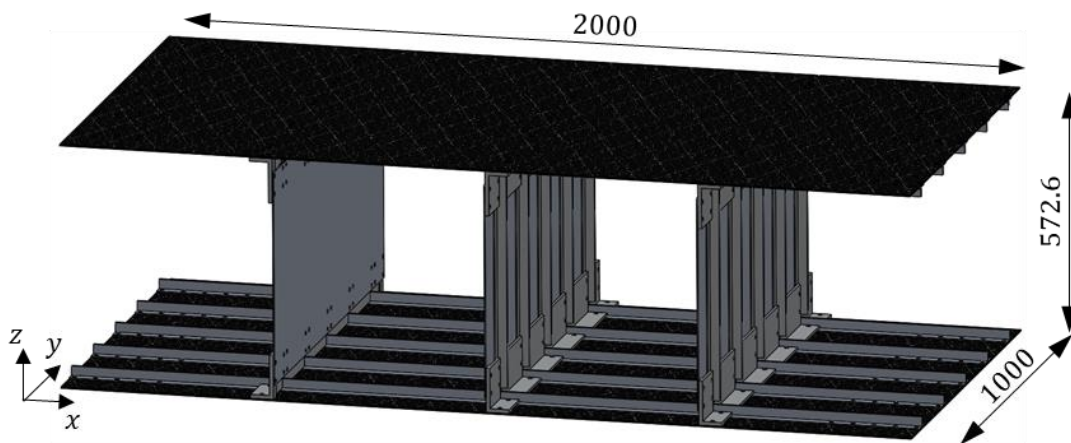


**Figure 1.1** – Wing representative structure. Drawing by School of Engineering, Cardiff University.

The structure is composed by aluminium and composite materials and is assembled with hybrid joints. All the materials, structural elements and joints resemble the ones used in a true wing structure and its fragilities and limitations are similar to the ones from the real structure, but the developed structure is not an actual airplane's wing.

### 1.3. OBJETIVES

To help with the goal of WG1 of performing an optimisation for an SMH system, the main objective of the present thesis is to study thoroughly the wing representative structure. Namely, its purpose is to identify probable damage mechanisms and critical locations, where a type of damage is prone to occur, and investigate about the use of VSHM techniques.

To do this, a number of milestones must be achieved:

- Review already existing progressive damage modelling and nonlinear VSHM methodologies;
- Perform a static analysis on the whole structure and compare with results from other WG1 members and experimental tests;
- Identify crucial points and/or components from the static test and select a simpler substructure with resembling problems to undergo the progressive damage and nonlinear dynamic analyses;
- Formulate and implement in ABAQUS software numerical models for the progressive damage and nonlinear dynamic analyses;
- Perform the progressive damage analysis of the identified substructure and conclude about the system's damage endurance;
- Perform the nonlinear dynamic analysis of the identified substructure and infer about the use of nonlinear VSHM technologies.

### 1.4. LAYOUT OF THE THESIS

The present thesis is divided into eight chapters, organized as follow:

- Chapter 1 – “Introduction”: in this chapter, the relevance of the present dissertation and its motivation are presented. This report's contextualisation in the COST action ODIN is explored and, finally, the principal objectives intended for the work are listed too;
- Chapter 2 – “State of the Art”: some of the topics of the thesis such as progressive damage modelling techniques and vibration-based structural health monitoring are reviewed, while mentioning, and comparing the existing alternatives as well as their advantages and disadvantages;
- Chapter 3 – “Mathematical Model”: in this section, the numerical model utilised is formulated and presented. Topics including Nonlinear Continuum and Contact Mechanics are approached;
- Chapter 4 – “Wing Representative Structure's Description”: the structure's geometry, constitution, and materials are explored. All mechanical properties needed, elastic, elastoplastic and damaged material ones, are listed. Micromechanics-based models are applied to define the properties of the composite material;
- Chapter 5 – “Wing Representative Structure's Analysis”: in this chapter, the wing representative structure undergoes static and modal analyses. The characteristics of these analyses are described in detail, including the FE model. From the results, a critical substructure is selected to be simulated in the next chapters;

- Chapter 6 – “Substructure’s Progressive Damage Analysis”: throughout this chapter, the results of the progressive damage analysis to the identified substructure are presented and discussed. The purpose is to identify the location most prone for a damage occurrence;
- Chapter 7 – “Substructure’s Dynamic Analysis”: aiming to conclude about the use of vibration-based structural health monitoring techniques in structures similar to the studied substructure, a dynamic analysis is carried out;
- Chapter 8 – “Conclusion”: lastly, a compilation of the most important remarks is presented, alongside some suggestions for future work.

## CHAPTER 2. STATE OF THE ART

---

### 2.1. INTRODUCTION

In this chapter, the state of the art of progressive damage modelling techniques and nonlinear VSHM methodologies is exhibited in sections 2.2 and 2.3, respectively. The point is to list and explain the already existing methodologies and to establish comparisons between them, highlighting their perks and limitations.

### 2.2. PROGRESSIVE DAMAGE MODELLING

As composite materials and adhesive/hybrid joints are widely used in structural aeronautical applications, it is crucial to design them properly. Consequently, several theories have emerged to increase their mechanical and structural performance.

Next, some of these theories are presented: first, the classical prediction ones, in section 2.2.1, and later, the most currently used methods, in section 2.2.2.

#### 2.2.1. CLASSICAL PREDICTION METHODS

The classical prediction methodologies are two: the strength of materials theory and the fracture mechanics approach.

In the first approach, materials are assumed to be free of defects and failure occurs when the stress or strain criteria is met. For instance, regarding bonded joints' failure, the maximum principal stress and strain criteria were used to predict the failure of single lap joints (Harris and Adams, 1984) and other geometries (Martiny et al., 2013). The von Mises criterion was successfully used in double lap joints by Charalambides et al. (1997) and the maximum shear strain one in single lap tubular joints by Lee and Lee (1992). In composite materials' failure, a bigger range of criteria can be applied: Choi et al. (1991) applied a quadratic average stress criterion on a laminated composite subjected to low-velocity impact; Hart-Smith (1998) and Bobrov et al. (2016) used the maximum stress criterion in composite laminates in biaxial and uniaxial loadings, respectively; Naik and Yerramalli (1998) and Mkaddem et al. (2008) choose the Tsai-Hill criterion for laminated composites under low velocity impact and plane stress loading, respectively; Hu et al. (2015) employed the Tsai-Wu criterion on fibre reinforced plastic composites when submitted to biaxial loads; Guillaumat and Hamdoun (2006) and Isbilir and Ghassemieh (2013) used the Hashin criterion for carbon fibre reinforced composite laminates under drilling operations.

However, the stress-based methods have trouble in cases of stress singularity, that is, situations where there is (at least) one point with infinite value of stress. This can happen near to any stress

inducing geometry such as notches or flaws in composite materials (Kashtalyan, 2010) or sharp corners in adhesive joints (da Silva and Öchsner, 2008). The problem can even be amplified by mesh refinement during a numerical analysis and lead to convergence issues (da Silva and Öchsner, 2008). To solve it, Wilson and Pipes (1981) used the point stress criterion where the stresses are evaluated in a single point at a characteristic distance and Nuismer and Labor (1978, 1979) applied the average stress method where stresses are averaged over a distance. For both methods, when the new stress values obtained reach the material strength, failure occurs. These methods surpass the stress singularity problem, but depend on the experimental determination of the characteristic distance and, therefore, cannot be extrapolated for different materials, stacking sequences or geometries (Kashtalyan, 2010).

Now, the fracture mechanics approach, which can be grounded in stress intensity factors or energetic concepts, considers the existence of defects in the material because of the manufacturing process or its handling. In other words, it is assumed that an initial crack or defect exists. This may be a downside of the method if the damage location is not evident.

In the stress intensity factors method, the crack propagates when the stress intensity factor,  $K$ , reaches the critical value, known as the material's fracture toughness,  $K_C$ . (da Silva and Öchsner, 2008). This concept was used by Pinninti and Ramamurthy.G (1997) for an orthotropic composite disk and by Moreira et al. (2007) in a cracked riveted lap joint. The stress intensity factors have the disadvantage of being local parameters and, because of that, they can be affected by the material's heterogeneity (Kashtalyan, 2010).

On the other hand, the energetic criterion presents itself as a global method and can overcome this problem. In it, the crack increases when the energy available at the crack tip, the so-called mode  $i$  strain energy release rate ( $G_i$ ), and due to the applied loading, exceeds a critical value ( $G_{iC}$ ), generally assumed as a material property. These strain energy release rates are commonly obtained by using the Virtual Crack Closure Technique, e.g. the work of Marjanović et al. (2016) in the delamination of laminated composite plates. This is the most used concept since the strain energy release rates have a physical meaning related with energy absorption and the stress intensity factors can be hard to determine (especially when the crack grows at or near an interface) (da Silva and Öchsner, 2008).

Both stress intensity factors and strain energy release rates are usually calculated for each crack propagation mode: mode I is an open mode and modes II and III are shear modes (Kashtalyan, 2010). However, in real life situations, as propagation can happen in several modes simultaneously, a mixed-mode criterion must be applied. A linear or quadratic one can be used as suggested in da Silva and Öchsner (2008).

Thus, to take the most of each method, current models combine them and predict damage initiation with stress criteria from the strength of materials method and damage propagation with fracture mechanics' concepts. This eliminates the initial crack requirement and minimizes mesh dependency problems.

### 2.2.2. CURRENT PREDICTION METHODS

The current methodologies to predict damage are the cohesive damage models and the continuum damage mechanics approach.

The cohesive damage models are a powerful tool usually based on softening relationships between stresses and relative displacements. With these, it is possible to simulate the loss of mechanical properties (da Silva and Öchsner, 2008).

Commonly, spring (Lammerant and Verpoest, 1996; Xie and Waas, 2006) or interface (de Moura et al., 2000; Camanho et al., 2003) finite elements (FE), without thickness, are applied at the planes where damage is likely to occur. This can, in many situations, be a restricting factor, because these planes can be hard to identify. However, in situations of bonded joints and composites' delamination, damage locus is easily predictable. In bonded joints, damage occurs in planes near or at the interface adhesive-adherend or inside the adhesive (da Silva and Öchsner, 2008). In the delamination of composites, it is restricted to the interface plane between two differently oriented layers (Kashtalyan, 2010).

What allows us to make a distinction between cohesive (or softening) damage models is the different traction-separation laws at their root. For instance, Gonçalves et al. (2003) and Kashtalyan (2010) applied triangular laws for damage prediction in brittle or moderately ductile adhesives. On the other hand, Dourado et al. (2012) and de Moraes et al. (2015) used bilinear cohesive laws in composites with two different and independent energy dissipation mechanisms. Blackman et al. (2003) employed a polynomial law in fibre composites and adhesively-bonded joints. Cousigné et al. (2013) used an exponential cohesive law in composites with nonlinear damage. Yang et al. (1999) and da Silva and Öchsner (2008) applied a trapezoidal law to ductile adhesives, with plasticity.

All models consist of two stages: an initial undamaged linear elastic stage and a subsequent softening damage process. In the first stage, stresses and relative displacements are proportional. The coefficient of proportionality is the penalty stiffness. This constant should have high values in order to avoid undesirable interpenetrations and numerical instabilities (Dourado et al., 2012). The second stage is what distinguishes each model since they have different formulations. It represents the softening of the material's properties, that is, the deterioration of the material's stiffness by dissipative processes like micro-cracking, fibre bridging and inelastic processes in composites (Kashtalyan, 2010) and plasticity in bonded joints (da Silva and Öchsner, 2008). In the simulation, failure occurs gradually mimicking what happens in the Fracture Process Zone (FPZ). Numerically, a damage parameter is implemented varying between zero (undamaged) and one (complete loss of stiffness) (da Silva and Öchsner, 2008).

To account for mixed-mode loadings, criteria must be applied to determine damage onset and propagation. Damage initiation can be determined by a stress based criterion, like the maximum stress (Vasiliev and Morozov, 2007) or the quadratic stress (Camanho and Dávila, 2002) criterion. On the other hand, damage propagation may be predicted by a fracture mechanics energetic criterion, e.g. in

Kashtalyan (2010) a linear fracture energetic criterion was applied and in Camanho and Dávila (2002) both a power law and the Benzeggagh-Kenane law were used.

Now, the appeal of continuum damage models lies in the fact that there is no need to know beforehand the damage location or path. However, these models are much more complex and can lead to numerical convergence problems. They are very useful in situations of thick adhesives, for example, when the adhesives thickness should not be neglected and when the propagation path might be non-symmetric (da Silva and Öchsner, 2008) or to simulate matrix cracking phenomenon (Kashtalyan, 2010). Unlike the previous ones, the continuum damage models in FE analysis use solid elements to simulate the degradation of the material's properties and are based on softening relationships between stresses and strains. To obtain these strains, a characteristic length must be considered, usually the same as the length of influence of a Gauss Point.

Similarly to cohesive models, the continuum ones are based on stress-strain laws, which may vary and also originate from multiple models. For instance, a triangular stress-strain law for pure modes is employed by de Moura and Chousal (2006) and an exponential one by Maimí et al. (2007). These models have two stages: an initial undamaged stage and a subsequent softening damage process. The first stage is characterised by a proportional stress-strain relation. The second stage, just like before, is what differentiates each model. According to it, the material properties at a given Gauss Point degrade gradually and mimic the energy release at the FPZ. Consequently, a load distribution for the neighbour points happens and the gradual propagation process is simulated. This approach has the upside of avoiding stress singularity effects and, thus, mesh sensitivity. Numerically, damage parameters like the previous ones are implemented.

### **2.3. VIBRATION-BASED STRUCTURAL HEALTH MONITORING**

Damage is defined as any unwanted change to material and/or geometric properties of a system, including changes in boundary conditions and system connectivity. This can adversely affect the current or future performance of the system (Worden et al., 2008). In the aeronautical and aerospace industries, this phenomenon is very important and can be the reason for a structure's failure. Therefore, damage detection methodologies are developing and evolving quickly. The main purposes are improving safety and optimising the cost/time relationship in the above-mentioned industries.

Traditionally, maintenance is based on statistical predictions of time between examinations. However, these are not optimal for cost savings. Moreover, since there has been a significant growth of the operational life of several airplanes, these costs, both direct (from repair costs) and indirect (from the imposed stop to the airplanes) are increasing (Carminati and Ricci, 2018).

Original damage detection approaches were non-destructive testing techniques and provided off-line and localised inspections, e.g. visual inspections, ultrasonic inspections, eddy currents, acoustic emissions, radiography, thermography or shearography (Bar-Cohen, 2000). Nowadays, interest relies on SHM technologies, particularly, on vibration-based structural health monitoring technologies. These allow an online inspection and are usually global methodologies, with potential for detecting

both global and local flaws in structures. Ideally, these methodologies should be sensitive to damage's initialisation and growth, as well as, they should be robust to noise and other factors like temperature and environmental conditions (Trendafilova and Manoach, 2012).

VSHM methods can be divided according to various criteria. According to its potential, the VSHM methods can be fitted in four levels: Level 1 – Damage confirmation; Level 2 – Damage identification and determination of the damage's location; Level 3 - Damage identification, determination of the damage's location and estimation of the damage severity; Level 4 – Damage impact evaluation on the structure and determination of remaining life (Manoach et al., 2017). Considering its type, VSHM models can be linear models assuming linear vibrational schemes or nonlinear models. Regarding data's processing, VSHM methods can be modal-based, where a comparison between modal parameters of the current structure with those of the undamaged one is established or response-based, where analyses of the results to proper dynamics tests are compared. Finally, paying attention to the data's origin, VSHM methods can be experimental-based, where the comparison is set between the experimental results, or model-based, where the comparison is between the results of experimental tests and numerical simulations (Carminati and Ricci, 2018).

Next, in the following sections the modal, response and model-based models are scrutinised since they are the most interesting for the developed analysis.

### **2.3.1. MODAL-BASED METHODS**

Looking closely at modal-based methods, these are based on the concept that damage causes a local reduction of stiffness and therefore a modification of modal parameters, e.g. natural frequencies, mode shapes, damping ratios and quantities derived from these parameters (Carminati and Ricci, 2018). In them, a fitting process to a physics-based model is done using data from the initial undamaged structure. Then, it is repeated periodically or after some potentially damaging event. Changes in the modal parameters indicate the presence (and sometimes the location) of damage (Worden et al., 2008).

As these are one of oldest VSHM methods, they have the advantage of being well defined and easy to implement. Furthermore, these approaches condensate the measured data in a simpler and more efficient way.

The natural frequencies are easily and accurately measured by many experimental approaches, but the analyses made with this feature lack sensitivity to damage (Andreaus and Baragatti, 2009; Manoach et al., 2012), since it is a local phenomenon and it does not significantly affect global stiffness. According to Tsyfansky and Beresnevich (2000), a damaged area of 15–20% on the cross section of a beam structure determines a reduction of the natural frequency of the element from 1–5%. Nevertheless, some authors established the relation between the presence and location of damage and the change of natural frequencies (Cawley and Adams, 1979).

As for the mode shapes, these are more sensitive to damage but are more difficult to obtain due to the need for many sensors to cover the structure's surface (if possible, because it could be

inaccessible). Great progress has been made to solve this problem by using contactless methods such as scanning laser Doppler vibrometer (Manoach et al., 2017).

The global character of modal methods can be transcended by increasing the amount of data to analyse (more sensors) and using data mining technologies such as artificial neural networks (ANN), principal component analysis (PCA) and genetic algorithms (GA) (Gordan et al., 2017; Carminati and Ricci, 2018).

It is also important to state that these methodologies have shown sensitivity to environmental conditions, e.g., temperature, humidity, or wind. The most important one is temperature: it changes the material stiffness, the boundary conditions and it induces thermal loads in the system, resulting in different modal properties. Thus, it can lead to false indications of damage or to no alarm when in fact there is a flaw (Trendafilova and Manoach, 2012). Besides, these approaches require initial undamaged data as a reference for comparison, that may not be available (Carminati and Ricci, 2018), and are usually based on linear physics models. However, most of the times, the portrayed structures present nonlinear dynamic behaviour. Examples of causes of nonlinearities are yielding, cracks or delaminations, rivets or bolts and frictions (Epureanu et al., 2004). All of these nonlinearities are aggravated by high amplitude structural vibrations (Trendafilova and Manoach, 2012).

### **2.3.2. RESPONSE-BASED METHODS**

Now, about the response-based methods, they are based on the comparison of the analysis of measured vibration signals from the undamaged and the damaged structures. If the original system is perfectly linear, the undamaged results may not even be needed. Their potential arises from their benefits: no physical model and, consequently, no linearity is assumed; a comparing model is expendable (if damage is the only existing nonlinearity); they are sensitive to small flaws, making them great to assess damage onset; and they are expected to be robust to noise and operational and/or environmental conditions, which is very relevant for the above-mentioned industries (Trendafilova and Manoach, 2012; Carminati and Ricci, 2018). Consequently, these approaches overcome many of the flaws presented by the modal-based methods. However, they are more complex and do not have such well-defined procedures.

One of the display methods for the collected data is a time-history, a graphical representation of time *versus*, e.g., displacement. The detection of damage in this method is possible by two ways: comparing the undamaged and the damaged time-history or comparing the harmonic distortion of the response with the input signal. The first way is pretty straightforward: it consists in overlapping the two responses and seeing if they disagree. In the harmonic distortion the purpose is to identify if the response is a mono-harmonic as the input (Worden et al., 2008). This second approach is based on the fact that a linear system's stationary response when excited with a mono-harmonic signal, such as a sine or cosine, is (also) a mono-harmonic at the same frequency (Worden et al., 2002). It is not always true to say that a mono-harmonic wave input to a nonlinear system does not produce a mono-harmonic wave output, yet this is the usual scenario. The appearance of higher-harmonics in the response is

expected. One advantage of using this methodology in experimental testing is that the measured results are, commonly, accelerations and these are very sensitive (compared with the corresponding velocity or displacement) to harmonic distortion due to the  $\omega^2$  factor weight (Worden et al., 2008).

Another way to represent the collected data is a phase portrait, a displacement *versus* velocity graphic. Phase space trajectories provide relevant information but can be very complex. The drawn trajectories seem to converge to an invariant subspace, the attractor. As this is a very ordered and simple result, its analysis is easy and attractor changes are interesting to explore. These methods are called attractor-based methods. There are two main ways to characterise attractors: using statistical methods or invariants. After having the attractor characterised, it is possible to establish a comparison between the results of the undamaged and the damaged structure and draw conclusions (Trendafilova and Manoach, 2012).

In the statistical approach, the method suggests the study of the statistical distribution of the attractor's points and uses it to extract damage sensitive features. For instance, the skewness and the variance have been shown sensitivity to damage and their relative changes have been used as damage features (Nichols et al., 2004; Trendafilova, 2006). These features can be used as local damage features, when their value is calculated based on only one point's results, or as global damage features, when data from multiple points in space is considered. Advantages of these methodologies are that they are easily applicable, can be built from results of any type of excitation, show sensitivity to damage and are robust to noise. As a downside, it is important to know that they rely on the data's quantity and quality.

The other approach uses invariants of the attractors. Examples of these are the Lyapunov exponents and the state space dimensions, whose relative changes can also be used as damage features. These methods show good sensitivity to damage, as well, while being simultaneously insensitive to noise. However, the invariant characteristics of the state space and the attractor are quite difficult to estimate from data and possess some variability/uncertainty (Trendafilova and Manoach, 2012).

An even simpler way to represent data is a Poincaré map. This map is often a projection of the phase portrait at moments in time  $t$ , equally spaced, so the Poincaré map is composed by a sequence of points. For a more precise definition, see Nayfeh and Balachandran (1994). Often,  $t$  is a multiple of the forcing periodic function's period. If a motion is characterised by a period  $n$ , there are  $n$  different points represented (Worden et al., 2008).

This method is very sensitive to any changes in the system, even if small. In addition, as there are few points to overlook, the analyses become much simpler. To profit from this, Manoach and Trendafilova (2008) have developed a damage index based on analysing the Poincaré sections. It is important to clarify that this analysis can be applied to multiple points of the studied structure and the damage index values will be larger for points near to the damaged zone, so, this can be used to localise damage (Trendafilova and Manoach, 2012).

Finally, one of the most dramatic manifestations of nonlinearity, chaos can also be applied in damage detection. As a curiosity, a chaotic regime is defined as one where there is a sensitive

dependence on initial conditions. As only nonlinear systems can behave chaotically, any feature that detects it, detects nonlinearities too and can identify damage. Examples of used features are Lyapunov exponents and the fractal attractor dimensions (Thompson and Stewart, 2001). Even though this approach is very useful in damage detection, it is suboptimal, since it does not increase monotonically with the damage extent (Worden et al., 2008).

Phase portraits and Poincaré representations are still used in this regime, but the results are much more complicated. The attractor of the phase space representation is now called a chaotic attractor. In addition, bifurcation diagrams are employed too. These are obtained by varying a system parameter, e.g., excitation frequency, and plotting it *versus* a one-dimensional projection of the Poincaré map associated with each value of the parameter, e.g., a point's displacement norm. In them, it is possible to observe the points at which the attractor's topology changes – bifurcation. These are helpful to identify the limited chaotic regions, as they cannot happen for all the excitation frequencies, where the above-mentioned features show nontrivial behaviour (Worden et al., 2008).

### **2.3.3. MODEL-BASED METHODS**

Model-methods can be seen as a complement to experimental ones and stand out in situations where there is not a healthy structure to compare results to. They also contribute to further developments in VSHM (Burlayenko and Sadowski, 2018), but have the downside of relying on experimental data for validation. Mostly, these methods can be based on analytical or numerical models, being FEM approaches preferred because of their versatility for solving complex topological and multi-physical problems (Burlayenko and Sadowski, 2018). Hereafter, a set of model-methods are presented and sorted by their increasing complexity.

A considerable amount of previous theoretical work has been carried out for studying the modal characteristics of delaminated beams, its natural frequencies, and modal shapes. In them, it is assumed that the detached surfaces at the delaminated interface do not interact with each other during vibrations and are freely vibrating (Wang et al., 1982; Kim and Hwang, 2002). Others have used the curvatures of mode shapes for quantifying the damage magnitude in a honeycomb beam (Lestari and Qiao, 2005) and a linear dynamic analysis is used in frequency response function-based for damage detection (Maia et al., 2003). However, all these theoretical methodologies are only applied on one-dimensional models.

When using FEMs, the restrictions are less. For example, in Cho and Kim (2001) a two-dimensional FE model was developed for laminated composite plates with multiple delaminations applying the higher-order zig-zag theory and the natural frequencies of the system were compared to assess damage. In Hu et al. (2002) and Kim et al. (2003), two-dimensional models for composite plates were also developed employing the higher-order shear deformation and the layerwise theories. Besides, in these analyses, the mode shapes were also observed. Finally, in Burlayenko and Sadowski (2012a) a three-dimensional model for a debonded sandwich panel was established and the natural

frequencies were evaluated. It is important to mention that all these approaches do not take into account the contact between the detached segments.

To solve the interpenetration problem between the detached parts fictitious linear springs can be introduced in the flaw. When the flaw is open, the spring stiffness is considered to be zero, and when it is closed, the contact stiffness is added to the equation of motion. Authors explored this concept in a theoretical way in Luo and Hanagud (2000); Tsyfansky and Beresnevich (2000) and Wang and Tong (2002) employed a three-dimensional FE models in Burlayenko and Sadowski (2010, 2011a). Despite preventing the interpenetration problem, this methodology does not present the interactions between the detached parts.

One of the possibilities to tackle contact problems is the use of the analytical approach, replacing a continuous cracked structure with a single or multi-degree of freedom oscillator. For instance, such spring-mass-damper models were successful to reveal nonlinear responses of composite beams with interlaminar damage in Chati et al. (1997); Chondros et al. (2001); Ajibose et al. (2010) and Li et al. (2011). Although the oscillator models used in those studies have enlightened the nonlinear phenomena of damaged structures, they lack information about the spatial presentation of both the deformation modes and distributions of interlaminar stresses within the flawed region. Thereby continuous models of vibrating structures damaged by cracks, explicitly describing the contact interaction between the detached segments, are needed.

More recently, several authors have dedicated their attention to the contact problem. In Žak et al. (2001); Kwon (2002) and Song and Belytschko (2009) node-to-node frictionless contact formulation have been explored and these constraints were imposed by a modified Lagrange multiplier or a penalty method. Others used a surface-to-surface contact definition in an explicit dynamic analysis (Burlayenko and Sadowski, 2011b, 2012b). In Burlayenko and Sadowski (2012a) an implicit analysis was developed with friction consideration and in Elmalich and Rabinovitch (2015) and Marjanović et al. (2015) besides the contact imposition, geometrical nonlinearities were also taken into account. In Motamedi and Mohammadi (2010) a crack growth problem with contact issues was solved using extended FEM.

Lastly, it is important to highlight the work of Burlayenko and Sadowski (2014) and Carminati and Ricci (2018) that are really relevant to the present report. Their importance is related with the dynamic analysis performed and geometric model used, respectively, which are very similar to the ones intended to be implemented here.

In Burlayenko and Sadowski (2014), a nonlinear dynamic analysis of a rectangular sandwich plate with a central penny-shaped debonded zone subjected to impulsive and harmonic loading was executed using a FE approach in ABAQUS software. The contact interface modelling was imposed by a penalty contact algorithm being that the impenetrability constraints were formulated by the Karush–Kuhn–Tucker (KKT) conditions and the friction law was the Coulomb friction model. Furthermore, Rayleigh’s damping was considered, and the implicit integration technique used was the Hilber–Hughes–Taylor (HHT) temporal integrator. The evaluated parameters were, in the impulsive analysis,

the natural frequencies and, in the harmonic one, the frequency response curves, the phase portraits and the Poincaré sections extracted from the time history signals.

At Carminati and Ricci (2018), a flat rectangular plate that reproduces a typical aeronautical structure is analysed. The structure is edged by two transversal supports with a rectangular section and five T-shaped stiffeners equally spaced to reinforce the longitudinal direction. The whole panel is built with aluminium 2024. Three damaged scenarios are recreated by the absence of connections between one stiffener and the base plate and are subjected to numerical and experimental analysis. Firstly, in the numerical analysis using MSC Nastran software the natural frequencies and the mode shapes are calculated assuming that the parts are freely vibrating since the nodes are unmerged. Next, in both analysis (and for each damage scenario) double sine excitation tests, tests for super harmonics and decay tests are performed.

## **2.4. FINAL REMARKS**

In this chapter, the state of the art of progressive damage modelling and nonlinear VSHM methodologies is presented and discussed.

Regarding the progressive damage modelling methods, the classical prediction ones are clearly outdated as they present limitations such as stress singularity problems, demand of an initial crack and mesh dependency problems. For this reason, the current models take the most of each classical method and combine them. Comparing the cohesive with the continuum damage model, the first is much simpler and has a lower computational cost but the second allows to simulate situations where the damage location is not known beforehand.

Hereafter, the employed progressive damage model is the cohesive damage one since the damage mechanisms explored are situations where the damage locus is easily predictable: composites' delamination and debonding in adhesive joints.

Concerning the nonlinear VSHM methods, the modal-based ones are the easiest to implement but require initial undamaged data and are usually based on linear physics models. On the other hand, the response-based ones do not rely on a physical model and a comparing model is unnecessary. However, they are more complex. Finally, the model-based methods stand out when there is not a sane structure to compare results to, but experimental validation is required.

Later on this thesis, a model-based approach is chosen, and a model of the studied structure is implemented in ABAQUS software. Also, to evaluate the acquired results, modal and response-based strategies are applied: comparison of natural frequencies, mode shapes, time-histories, phase portraits, Poincaré sections and frequency spectra.

## CHAPTER 3. MATHEMATICAL MODEL

---

### 3.1. INTRODUCTION

In this chapter, the model utilised for the numerical analysis is formulated and presented. Firstly, the theoretical foundations of general nonlinear continuum mechanics are presented as a way to introduce central definitions and notation. Second, building upon the continuum mechanics concepts, topics of general contact mechanics theory, such as contact kinematics and normal contact constraints, are summarised. Next, the weak form is laid down, preparing grounds for the FE discretisation. Lastly, the derivation of the complete discrete version of the weak form and the application of the finite element method (FEM) are introduced.

### 3.2. CONTINUUM MECHANICS

Continuum mechanics (or nonlinear solid mechanics) perceives the mechanical behaviour of materials as a continuous mass model instead of discrete particles. In such a study, the assumption of continuously distributed matter, without gaps or empty spaces is made. Thus, material is seen as a continuous medium: an infinite set of particles that are studied macroscopically, namely, without considering the possible discontinuities existing at an atomic or molecular level. Accordingly, one admits that there are no discontinuities between the particles and that the mathematical description of this medium and its properties can be described by continuous functions.

Throughout this chapter, topics such as deformation kinematics description, strain and stress measures, governing equations and general aspects of constitutive models are addressed. For an in-depth overview of nonlinear continuum mechanics, readers are referred to the textbooks by Holzapfel (2002); Bonet and Wood (2008); Kim (2015); Korelc and Wriggers (2016) and Oliver and Saracibar (2017).

#### 3.2.1. KINEMATICS

The kinematical relations concern the study of motion and deformation of a continuum body, the derivation of strain measures and the time derivatives of kinematical quantities. Therefore, the motion of a body along space and time can be defined.

A continuum body  $\mathcal{B}$  composed of a set of particles  $\mathcal{P} \in \mathcal{B}$  is a continuous distribution of matter in time and space. This body is embedded in three-dimensional Euclidean space  $\mathbb{R}^3$ . The continuum occupies different regions through time  $t$  that are called configurations  $\Omega \in \mathbb{R}^3$  – Figure 3.1.

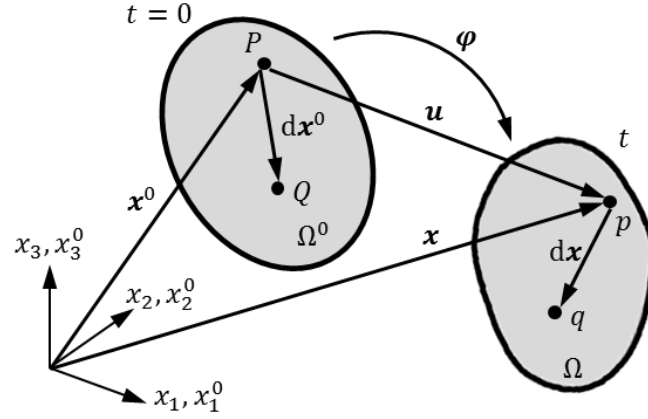


**Figure 3.1** – Motion of a continuum body  $\mathcal{B}$  from the initial to the current configurations. Adapted from Reddy (2013).

In a Lagrangian perspective, for the reference time  $t = 0$ , the configuration at this time is  $\Omega^0$  and is termed as the initial, material or reference configuration. Considering a Cartesian coordinate system  $(X, Y, Z)$ , the position vector (or the material coordinates) of a particle  $P \in \Omega^0$  is  $\mathbf{x}^0$ .

When some external stimuli are applied to the system, a new configuration  $\Omega$  is occupied at a given time  $t$ . The motion represented by the vector field  $\boldsymbol{\varphi}$  is used to do the deformation mapping of the body  $\mathcal{B}$  from  $\Omega^0$  to  $\Omega$  and relates the current configuration  $\mathbf{x}$  with the reference configuration  $\mathbf{x}^0$  for each time  $t$ . This field is a bijective nonlinear function. So, the position vector of particle  $p \in \Omega$  and the original position vector are

$$\mathbf{x} = \boldsymbol{\varphi}(\mathbf{x}^0, t), \quad (3.1)$$

$$\mathbf{x}^0 = \boldsymbol{\varphi}^{-1}(\mathbf{x}, t). \quad (3.2)$$

The displacement field of particle  $P$  (also represented in Figure 3.1) may be described using both descriptions. Respectively, the Lagrangian and the Eulerian descriptions are

$$\mathbf{u}^0(\mathbf{x}^0, t) = \mathbf{x}(\mathbf{x}^0, t) - \mathbf{x}^0, \quad (3.3)$$

$$\mathbf{u}(\mathbf{x}, t) = \mathbf{x} - \mathbf{x}^0(\mathbf{x}, t). \quad (3.4)$$

The dependence of the deformation on time  $t$  must be considered in nonlinear problems in case that the constitutive behaviour is history dependent, like in plasticity, or in case of a dynamic process.

The velocity of a particle  $\dot{\mathbf{x}}$  is the first derivative of the motion  $\boldsymbol{\varphi}(\mathbf{x}^0, t)$  with respect to time  $t$ . Even though it is expressed in terms of the material coordinates, velocity is a spatial field. Its representation in material and spatial coordinates is

$$\dot{\mathbf{x}}(\mathbf{x}^0, t) = \frac{\partial \boldsymbol{\varphi}(\mathbf{x}^0, t)}{\partial t}, \quad (3.5)$$

$$\dot{\mathbf{x}}(\mathbf{x}, t) = \dot{\mathbf{x}}(\boldsymbol{\varphi}^{-1}(\mathbf{x}, t), t). \quad (3.6)$$

On the other hand, the acceleration of a particle  $\ddot{\mathbf{x}}$  is the second derivative of the motion  $\boldsymbol{\varphi}(\mathbf{x}^0, t)$  in respect to time  $t$  and it is represented in material and spatial coordinates is, respectively,

$$\ddot{\mathbf{x}}(\mathbf{x}^0, t) = \frac{\partial^2 \boldsymbol{\varphi}(\mathbf{x}^0, t)}{\partial t^2}, \quad (3.7)$$

$$\ddot{\mathbf{x}}(\mathbf{x}, t) = \ddot{\mathbf{x}}(\boldsymbol{\varphi}^{-1}(\mathbf{x}, t), t). \quad (3.8)$$

Another important concept to define is the spatial velocity gradient  $\mathbf{l}$  which corresponds to the derivative of the velocity field with respect to the spatial coordinates (considering  $\nabla_{\mathbf{x}}$  as the spatial gradient operator)

$$\mathbf{l} = \nabla_{\mathbf{x}} \dot{\mathbf{x}} = \frac{\partial \dot{\mathbf{x}}}{\partial \mathbf{x}}. \quad (3.9)$$

This gradient can be divided in a symmetric part  $\mathbf{d} = \text{symm } \mathbf{l}$ , named the rate of deformation tensor, and a skew part  $\mathbf{w} = \text{skew } \mathbf{l}$ , called the spin tensor.

### 3.2.2. STRAIN MEASURES

The deformation to which the body  $\mathcal{B}$  is submitted may be described by the deformation gradient  $\mathbf{F}$ . This tensor provides the relation between the material lines  $d\mathbf{x}^0$  and  $d\mathbf{x}$  before and after deformation

$$d\mathbf{x} = \mathbf{F}(\mathbf{x}^0, t) d\mathbf{x}^0. \quad (3.10)$$

These lines are also schematised in Figure 3.1. Using a material gradient operator  $\nabla_{\mathbf{x}^0}$ , the deformation gradient is defined by

$$\mathbf{F}(\mathbf{x}^0, t) = \frac{\partial \boldsymbol{\varphi}(\mathbf{x}^0, t)}{\partial \mathbf{x}^0} = \nabla_{\mathbf{x}^0} \boldsymbol{\varphi}(\mathbf{x}^0, t). \quad (3.11)$$

This equation is set assuming that the body is subjected to an inhomogeneous deformation and thus  $\mathbf{F} = \mathbf{F}(\mathbf{x}^0, t)$ . If the deformation is homogeneous then  $\mathbf{F} = \mathbf{F}(t)$ .

The gradient may still be written as a function of the displacement field, such as

$$\mathbf{F}(\mathbf{u}^0, t) = \nabla_{\mathbf{x}^0} \mathbf{u}^0 + \mathbf{I}, \quad (3.12)$$

$$\mathbf{F}^{-1}(\mathbf{u}, t) = \nabla_{\mathbf{x}} \mathbf{u} - \mathbf{I}, \quad (3.13)$$

where  $\mathbf{I}$  is the identity matrix or second-order tensor.

The Jacobian determinant  $J$  is the determinant of the deformation gradient

$$J = \det \mathbf{F}. \quad (3.14)$$

It represents the volume change, and it cannot be  $J \leq 0$ , since  $\mathbf{F}$  has no singularity and self-penetration is not possible. Also, if  $J = 1$  that means that the deformation is isochoric.

After knowing the definition of the deformation gradient and its determinant, it is possible to establish the transformation of the surface ( $d\mathbf{a}^0$ ) and volume ( $d\nu^0$ ) elements in the material

configuration to the spatial one ( $d\mathbf{a}$ ,  $dv$ )

$$d\mathbf{a} = J\mathbf{F}^{-T}d\mathbf{a}^0, \quad (3.15)$$

$$dv = Jdv^0. \quad (3.16)$$

According to the polar decomposition theorem, any deformation can be split into a rigid body rotation, portrayed by a rotation tensor  $\mathbf{R}$ , and a distortion, represented by a right  $\mathbf{U}$  or left  $\mathbf{V}$  stretch tensor. The deformation field can be rewritten as  $\mathbf{F} = \mathbf{R}\mathbf{U} = \mathbf{V}\mathbf{R}$ .

Since the deformation gradient is not symmetric, another measure of deformation may be set by the right Cauchy-Green deformation tensor  $\mathbf{C}$  in the material coordinates

$$\mathbf{C} = \mathbf{F}^T\mathbf{F} = \mathbf{U}^T\mathbf{U}, \quad (3.17)$$

and the left Cauchy-Green deformation tensor  $\mathbf{b}$  in the spatial coordinates

$$\mathbf{b} = \mathbf{F}\mathbf{F}^T = \mathbf{V}\mathbf{V}^T. \quad (3.18)$$

These have the advantage of being symmetric which favours the development of the model.

Strains measures evaluate changes in distance and angles between particles, that is, evaluate if the body has suffered any stretch and/or rotation between its particles. They are obtained by deriving the displacements to the reference coordinates. Using a material coordinate system, the Green-Lagrange strain tensor  $\mathbf{E}$  is defined by

$$\mathbf{E}(\mathbf{x}^0, t) = \frac{1}{2}(\nabla_{\mathbf{x}^0}\mathbf{u} + \nabla_{\mathbf{x}^0}\mathbf{u}^T - \nabla_{\mathbf{x}^0}\mathbf{u}^T \nabla_{\mathbf{x}^0}\mathbf{u}) = \frac{1}{2}(\mathbf{C} - \mathbf{I}), \quad (3.19)$$

and, with a spatial one, the Euler-Almansi strain tensor  $\mathbf{e}$  is

$$\mathbf{e}(\mathbf{x}, t) = \frac{1}{2}(\nabla_{\mathbf{x}}\mathbf{u} + \nabla_{\mathbf{x}}\mathbf{u}^T - \nabla_{\mathbf{x}}\mathbf{u}^T \nabla_{\mathbf{x}}\mathbf{u}) = \frac{1}{2}(\mathbf{I} - \mathbf{b}^{-1}). \quad (3.20)$$

Both Green-Lagrange and Euler-Almansi strain tensors are strain measures independent of the body's rotation and should be used when a body has large displacements. However, in situations of small displacement gradients, the infinitesimal strain tensor  $\boldsymbol{\varepsilon}$  defined as

$$\boldsymbol{\varepsilon} = \frac{1}{2}(\nabla_{\mathbf{x}^0}\mathbf{u} + \nabla_{\mathbf{x}^0}\mathbf{u}^T) = \frac{1}{2}(\nabla_{\mathbf{x}}\mathbf{u} + \nabla_{\mathbf{x}}\mathbf{u}^T), \quad (3.21)$$

can be used. This tensor has the advantage of neglecting the nonlinear terms of the previous strain tensors and, thus, varying linearly with the displacement gradient.

Finally, the material strain rate tensor  $\dot{\mathbf{E}}$  is the derivative with respect to time of the Lagrangian strain tensor

$$\dot{\mathbf{E}} = \frac{1}{2}\dot{\mathbf{C}} = \frac{1}{2}(\dot{\mathbf{F}}^T\mathbf{F} + \mathbf{F}^T\dot{\mathbf{F}}), \quad (3.22)$$

where  $\dot{\mathbf{F}}$  is the time derivative of the deformation gradient.

### 3.2.3. STRESS MEASURES

Motion and deformation originate interactions between the material and neighbouring material in the interior part of the body. One consequence of these interactions is stress. It has physical dimension force per unit of area, is responsible for the deformation of materials and crucial in continuum mechanics.

In large displacement analysis, the used coordinate system is important. Considering an area element  $\Delta a$ , with an outward normal  $\mathbf{n}$  in the neighbourhood spatial point  $p$  – Figure 3.2. For a resultant force acting upon this area element of  $\Delta \mathbf{f}$ , it is possible to define the traction vector  $\mathbf{t}$  as

$$\mathbf{t}(\mathbf{x}, t, \mathbf{n}) = \lim_{\Delta a \rightarrow 0} \frac{\Delta \mathbf{f}}{\Delta a} = \boldsymbol{\sigma} \mathbf{n}, \quad (3.23)$$

where  $\boldsymbol{\sigma}$  is the Cauchy stress tensor (also named the true stress tensor). This tensor is symmetric with six independent components, meaning that  $\boldsymbol{\sigma} = \boldsymbol{\sigma}^T$ .

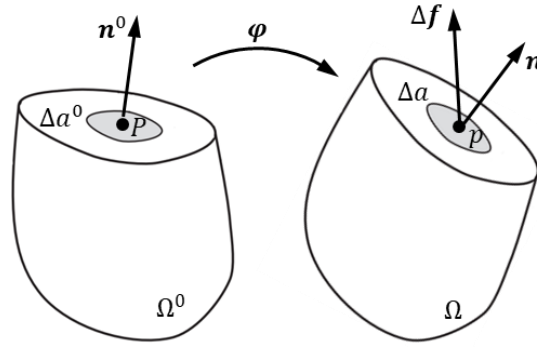


**Figure 3.2** – Stress vectors in the initial and current configurations. Adapted from Bonet and Wood (2008).

Other stress tensor used is the Kirchhoff stress tensor that is related to the Cauchy stress one by

$$\boldsymbol{\tau} = J \boldsymbol{\sigma}. \quad (3.24)$$

Yet, in large displacement scenarios, using stress measures referent to the initial configuration can be more effective. Similarly, it is possible to calculate the traction vector  $\mathbf{t}^0$  in the undeformed configuration as

$$\mathbf{t}^0(\mathbf{x}^0, t, \mathbf{n}^0) = \lim_{\Delta a^0 \rightarrow 0} \frac{\Delta \mathbf{f}}{\Delta a^0} = \mathbf{P}^T \mathbf{n}^0. \quad (3.25)$$

$\mathbf{P}$  is the first Piola-Kirchhoff stress tensor which is an asymmetric second order tensor referred to the material configuration. It is possible to relate it with the Cauchy stress tensor by

$$\mathbf{P} = J \boldsymbol{\sigma} \mathbf{F}^{-1}. \quad (3.26)$$

To obtain a symmetric tensor, the second Piola-Kirchhoff stress tensor  $\mathbf{S}$  is introduced as

$$\mathbf{S} = \mathbf{F}^{-1} J \boldsymbol{\sigma} \mathbf{F}^{-1}. \quad (3.27)$$

### 3.2.4. GOVERNING EQUATIONS

Continuum Mechanics is based on a series of general postulates or principles that are assumed to be always valid, regardless of the type of material and the range of displacements or deformations. Among these are the so-called balance principles: balance of mass, balance of linear momentum and balance of angular momentum. In this section, the spatial differential formulation of these principles is presented.

The balance of mass states that the mass of a continuous medium is always the same. That means that the mass is the same in the undeformed  $\Omega^0$  and in the deformed  $\Omega$  configurations

$$\rho \operatorname{div}_x \dot{\mathbf{x}} + \dot{\rho} = 0 . \quad (3.28)$$

In this equation the variable  $\dot{\rho}$  stands for the mass flux that crosses the control volume.

In the balance of linear momentum, the resultant of all the forces acting on a material volume of the continuous medium is equal to the variation of its linear momentum

$$\operatorname{div}_x \boldsymbol{\sigma} + \mathbf{b} = \rho \ddot{\mathbf{x}} . \quad (3.29)$$

It is considered that there are two types of external forces acting on the continuous body: surface tractions  $\mathbf{t}$  acting in the body surface  $\partial\Omega$  and volume forces  $\mathbf{b}$  acting in its volume  $\Omega$ .  $\rho \ddot{\mathbf{x}}$  are the symmetric of inertial forces which can be disregarded in static problems.

The balance of angular momentum states that the variation of angular momentum with respect to a point in space is equal to the moments resulting from the external forces acting on the body with respect to that same point. This results in the Cauchy's stress tensor symmetry  $\boldsymbol{\sigma} = \boldsymbol{\sigma}^T$ .

### 3.2.5. CONSTITUTIVE LAWS

The equations introduced so far characterise kinematics, strains and stresses and are applicable to any continuum irrespective of its physical constitution. These equations are not sufficient on their own to determine the material response, therefore, additional equations in the form of constitutive laws must be established.

Constitutive equations relate primary field, e.g.,  $\mathbf{u}$ , to secondary field variables, e.g.,  $\boldsymbol{\sigma}$ , and implement the intrinsic physical properties of a continuum. They are essentially mathematical models of the real behaviour of materials that are validated against experimental results.

The purpose of this chapter is to display the phenomenological constitutive equations used throughout this report: elasticity and elastoplasticity equations and damage equations for cohesive materials.

Reference literature for elastic behaviour are Holzapfel (2002); Popp (2012) and Kim (2015), for elastoplasticity Bonet and Wood (2008); Korelc and Wriggers (2016) and Oliver and Saracibar (2017) and for damage evaluation Camanho et al. (2003); da Silva and Öchsner (2008), Kashtalyan (2010) and Dourado et al. (2012).

### Elastic Materials

Objects deform when loads are applied to them. However, if the material is elastic, after the load's removal, it has the ability of returning to its initial undeformed configuration.

In a general point of view, the constitutive relation that describes this phenomenon is a nonlinear one suitable to describe a wide variety of physical phenomena in which the strains may be large. This theory is called finite hyperelasticity.

In hyperelastic or Green-elastic materials the work done by the stresses during a deformation process is path-independent, that is, the work only depends on the initial and on the deformed configurations. As a consequence of this path-independent behaviour, a strain energy function or elastic potential  $\Psi$  (defined per unit of undeformed volume) which illustrates the elastic strain energy stored in the body may be defined.

Commonly, the second Piola-Kirchhoff stress tensor can be written based on the strain-energy function by the total Lagrangian constitutive equation as

$$\mathbf{S} = 2 \frac{\partial \Psi(\mathbf{C}, \mathbf{x}^0)}{\partial \mathbf{C}} = \frac{\partial \Psi(\mathbf{E}, \mathbf{x}^0)}{\partial \mathbf{E}}. \quad (3.30)$$

In situations of infinitesimal deformations, i.e.,  $\|\nabla_{\mathbf{x}} \mathbf{u}\| \ll 1$ , the nonlinear constitutive relation previously written becomes linear and the materials are now denominated linear elastic. In these conditions, the distinction between the various measures of stress and strain can be disregarded:  $\mathbf{S} \approx \boldsymbol{\sigma}$  and  $\mathbf{E} \approx \boldsymbol{\varepsilon}$  and the strain energy function is independent of the position vector:  $\Psi(\boldsymbol{\varepsilon})$ .

Therefore, the constitutive equation for linear elastic (or linearized hyperelastic) materials is

$$\boldsymbol{\sigma} = \frac{\partial \Psi(\boldsymbol{\varepsilon})}{\partial \boldsymbol{\varepsilon}}. \quad (3.31)$$

This relation can even be further simplified to the generalised Hooke's law

$$\boldsymbol{\sigma} = \mathbb{K} : \boldsymbol{\varepsilon}, \quad (3.32)$$

where  $\mathbb{K}$  is a fourth-order tensor containing the elastic stiffness coefficients  $K_{ijkl}$ .

Originally, there are 81 scalar components in this tensor. However, taking advantage of the symmetry conditions of these components and the stress and strain tensors, the number of independent coefficients  $K_{ijkl}$  drops to 21. As a side note, is important to mention that in engineering applications, the notation of these elastic stiffness coefficients is simplified to  $K_{ij}$ , where each index indicates the row  $i$  and column  $j$ , like

$$\mathbb{K} = \begin{bmatrix} K_{1111} & K_{1122} & K_{1133} & K_{1123} & K_{1113} & K_{1112} \\ & K_{2222} & K_{2233} & K_{2223} & K_{2213} & K_{2212} \\ & & K_{3333} & K_{3323} & K_{3313} & K_{3312} \\ & & & K_{2323} & K_{2313} & K_{2312} \\ & & & & K_{1313} & K_{1312} \\ & & & & & K_{1212} \end{bmatrix} = \begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\ & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\ & & K_{33} & K_{34} & K_{35} & K_{36} \\ & & & K_{44} & K_{45} & K_{46} \\ & & & & K_{55} & K_{56} \\ & & & & & K_{66} \end{bmatrix}. \quad (3.33)$$

For orthotropic, transversely isotropic, or isotropic materials, the number of independent coefficients is even lower, 9, 6 and 2, respectively. The fourth-order tensor containing the elastic stiffness coefficients for orthotropic and isotropic materials is presented in Equations (3.34) and (3.35), where  $E$  stands for Young's modulus,  $\nu$  for Poisson's ratio,  $G$  for shear modulus and coordinate system  $123$  for principal material directions:

$$\mathbb{K}_{ORTHO.} = \begin{bmatrix} \frac{1 - \nu_{23}\nu_{32}}{E_2 E_3 \Delta} & \frac{\nu_{21} + \nu_{23}\nu_{31}}{E_2 E_3 \Delta} & \frac{\nu_{31} + \nu_{21}\nu_{32}}{E_2 E_3 \Delta} & 0 & 0 & 0 \\ & \frac{1 - \nu_{23}\nu_{32}}{E_1 E_3 \Delta} & \frac{\nu_{32} + \nu_{12}\nu_{31}}{E_1 E_3 \Delta} & 0 & 0 & 0 \\ & & \frac{1 - \nu_{12}\nu_{21}}{E_1 E_2 \Delta} & 0 & 0 & 0 \\ & & & G_{23} & 0 & 0 \\ & & & & G_{13} & 0 \\ & & & & & G_{12} \end{bmatrix}, \quad (3.34)$$

$$\Delta = \frac{1 - \nu_{12}\nu_{21} - \nu_{23}\nu_{32} - \nu_{13}\nu_{31} - 2\nu_{21}\nu_{32}\nu_{13}}{E_1 E_2 E_3},$$

$$\mathbb{K}_{ISO.} = \frac{E}{(1 + \nu)(1 - 2\nu)} \begin{bmatrix} 1 - \nu & \nu & \nu & 0 & 0 & 0 \\ & 1 - \nu & \nu & 0 & 0 & 0 \\ & & 1 - \nu & 0 & 0 & 0 \\ & & & \frac{1 - 2\nu}{2} & 0 & 0 \\ & & & & \frac{1 - 2\nu}{2} & 0 \\ & & & & & \frac{1 - 2\nu}{2} \end{bmatrix}. \quad (3.35)$$

The fourth-order tensor for transversely isotropic materials is obtained from Equation (3.34) by matching  $E_2 = E_3$  and  $G_{12} = G_{13}$ .

### Elastoplastic Materials

Different from elastic materials, some materials acquire permanent deformation when a stress larger than an elastic limit is applied and can no longer be represented by simpler elastic models. This kind of nonlinear material behaviour is named plasticity. More precisely, elastoplasticity since these materials are initially elastic and then become plastic. This lack of elasticity is observed when the materials are freed from stresses, and fail to return to the initial undeformed configuration.

Contrary to the previously described hyperelastic behaviour, in elastoplastic materials stress calculation is history or path dependent.

An elemental vector  $d\mathbf{x}^0$ , belonging to the initial configuration, in the local neighbourhood of a given initial particle  $P$  deforms into a spatial vector  $d\mathbf{x}$  of the deformed configuration in the neighbourhood of  $p$ . If the neighbourhood of  $p$  could be isolated and freed from all forces, the material would achieve a new unloaded configuration characterised by the spatial vector  $d\tilde{\mathbf{x}}$ . This configuration is an overall stress-free plastic state. All these configurations are represented in Figure 3.3.

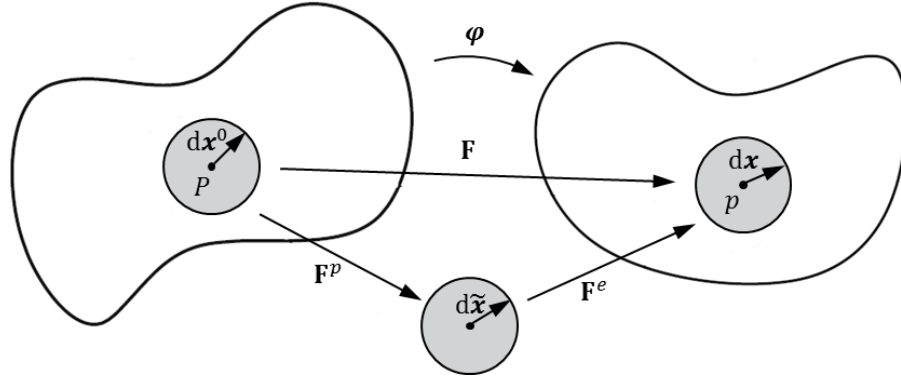


**Figure 3.3** – Multiplicative decomposition in the small neighbourhood of a particle. Adapted from Korelc and Wriggers (2016).

Using the multiplicative decomposition of the deformation gradient into elastic and permanent components ( $\mathbf{F}^e$  and  $\mathbf{F}^p$ , respectively), it is possible to write

$$\mathbf{F} = \frac{\partial \mathbf{x}}{\partial \tilde{\mathbf{x}}} \frac{\partial \tilde{\mathbf{x}}}{\partial \mathbf{x}^0} = \mathbf{F}^e \mathbf{F}^p . \quad (3.36)$$

Likewise, and since the right Cauchy-Green tensors are independent of rigid body rotations, they can be rewritten as

$$\tilde{\mathbf{C}}^e = \mathbf{F}^{eT} \mathbf{F}^e , \quad (3.37)$$

$$\mathbf{C}^p = \mathbf{F}^{pT} \mathbf{F}^p . \quad (3.38)$$

It is worthy of mention that the right plastic Cauchy-Green tensor  $\mathbf{C}^p$  is defined in the reference configuration  $\mathbf{x}^0$ , whereas the right elastic Cauchy-Green tensor  $\tilde{\mathbf{C}}^e$  is referenced to the permanent plastic state  $\tilde{\mathbf{x}}$ .

The strain energy function is now a function of the right elastic Cauchy-Green tensor and the position vector:  $\Psi = \Psi(\tilde{\mathbf{C}}^e, \mathbf{x}^0)$ . Just like in the hyperelastic regime, the second Piola-Kirchhoff stress tensor (now referenced to the permanent plastic state  $\tilde{\mathbf{x}}$ ) can be written based on the strain-energy function

$$\tilde{\mathbf{S}} = 2 \frac{\partial \Psi(\tilde{\mathbf{C}}^e, \mathbf{x}^0)}{\partial \tilde{\mathbf{C}}^e} . \quad (3.39)$$

From it, the Cauchy stress tensor may be achieved

$$\boldsymbol{\sigma} = \mathbf{F}^e \tilde{\mathbf{S}} \mathbf{F}^{eT} . \quad (3.40)$$

In the next equations, only rate-independent plasticity situations are considered. This means that the material's response does not depend on the rate of deformation  $\mathbf{d}$ . Also, throughout this report, a von Mises yield criterion is used, and the material's hardening is linear isotropic. All the elastoplastic materials considered are isotropic.

The yield function  $f$  is used to precisely define the elastic region and its limits. If  $f < 0$ , then the material behaves elastically, if  $f = 0$ , then the stress tensor must be such that the equivalent stress equals the current yield stress and elastoplastic behaviour ensues.

For the von Mises criterion, the yield function may be represented by

$$f(\boldsymbol{\sigma}, \alpha_p) = \sqrt{\frac{3}{2}} \|\mathbf{s}\| - \sigma_{ref}(\alpha_p) \leq 0. \quad (3.41)$$

The first term refers to the equivalent stress and the second one to the current yield stress  $\sigma_{ref}(\alpha_p)$  that varies with the hardening variable  $\alpha_p > 0$  and depends on the hardening law. Also,  $\mathbf{s}$  is the deviatoric component of the stress defined as

$$\mathbf{s} = \boldsymbol{\sigma} - p\mathbf{I}, \quad (3.42)$$

where  $p$  is the hydrostatic pressure.

Based on the yield criterion, a fundamental concept possible to define is the one of initial elastic domain  $\mathbb{E}^0$  which is

$$\mathbb{E}^0 = \{\boldsymbol{\sigma} \mid f(\boldsymbol{\sigma}, \alpha_p = 0) < 0\}, \quad (3.43)$$

and initial yield surface  $\partial\mathbb{E}^0$ ,

$$\partial\mathbb{E}^0 = \{\boldsymbol{\sigma} \mid f(\boldsymbol{\sigma}, \alpha_p = 0) = 0\}. \quad (3.44)$$

Both regions are represented in Figure 3.4.

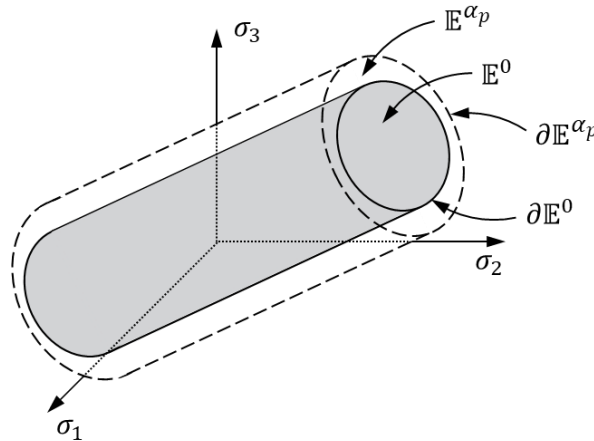


**Figure 3.4** – Initial and current elastic domains and yield surfaces. Adapted from Oliver and Saracibar (2017).

The hardening law used is described by

$$\sigma_{ref}(\alpha_p) = \sigma_y + A(\alpha_p) = \sigma_y + H\alpha_p, \quad (3.45)$$

where  $\sigma_y$  stands for initial yield stress (a material's characteristic) and  $H$  for the material-hardening parameter.

This rule results in the expansion of the initial elastic domain and yield surface to the current elastic domain  $\mathbb{E}^{\alpha_p}$  and the current yield surface  $\partial\mathbb{E}^{\alpha_p}$  (also presented in Figure 3.4).

Now, for elastoplastic materials, the rate of deformation is given by an equation known as the flow rule which relates it to the current state of stress. An associative flow rule, where the direction of the plastic strain rate coincides with the gradient of the yield surface, is assumed here.

Similarly to the deformation gradient, the rate of deformation tensor can also be divided according to its elastic and plastic components –  $\mathbf{d}^e$  and  $\mathbf{d}^p$ , respectively. The plastic rate of deformation gradient can be defined by

$$\mathbf{d}^p = \lambda \frac{\partial f(\boldsymbol{\sigma}, \alpha_p)}{\partial \boldsymbol{\sigma}} = \lambda \sqrt{\frac{3}{2}} \frac{\mathbf{s}}{\|\mathbf{s}\|}, \quad (3.46)$$

where  $\lambda$  is a proportionality factor called the consistency parameter or plastic multiplier.

The hardening rate  $\dot{\alpha}_p$  that defines the evolution of the hardening variable is

$$\dot{\alpha}_p = \lambda \frac{\partial f(\boldsymbol{\sigma}, \alpha_p)}{\partial A(\alpha_p)} = \lambda. \quad (3.47)$$

Finally, to obtain the plastic multiplier, the loading-unloading conditions and the consistency condition are added

$$\lambda \geq 0, \quad (3.48)$$

$$f(\boldsymbol{\sigma}, \alpha_p) \leq 0, \quad (3.49)$$

$$\lambda f(\boldsymbol{\sigma}, \alpha_p) = 0. \quad (3.50)$$

### Damaged Materials

The cohesive damage models are a powerful tool in the simulation of damage initiation and growth. They do not require an initial crack or flaw and are usually based on softening relationships between stresses and relative displacements. With these, it is possible to simulate progressive damage by the loss of the mechanical properties. Also, most of the times, stress-based criteria and energetic fracture concepts are chosen to simulate damage initiation and propagation, respectively.

The presented formulation is based on interface FE, without thickness, which are applied at the planes where damage is likely to occur. The surfaces where the cohesive elements are placed are denominated in the initial configuration by  $\Gamma_d^0$  and in the current one by  $\Gamma_d$ .

What allows us to make a distinction between cohesive damage models is the different traction-separation laws at their root. Through this chapter a triangular traction-separation law is explored – Figure 3.5. This type of model is applicable for brittle or moderately ductile adhesives.

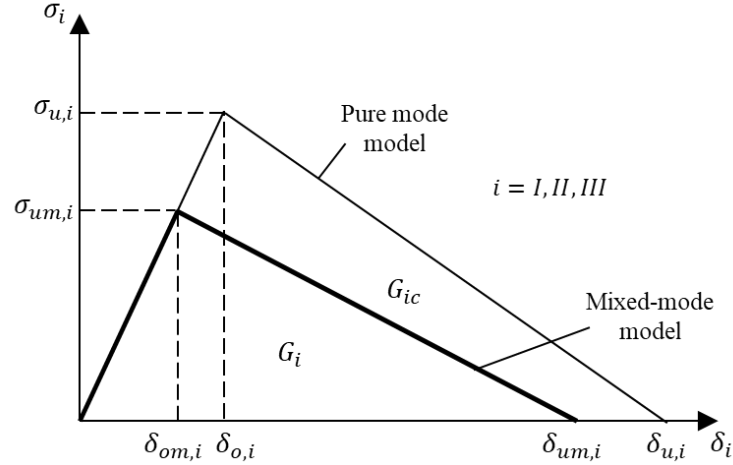


**Figure 3.5** – Triangular pure and mixed-mode cohesive damage model. Adapted from Kashtalyan (2010).

Common to all models is the fact that the area below the pure mode  $i$  model is the critical strain energy release rate ( $G_{ic}$ ). Also, is the fact that all models are constituted by two stages: an initial undamaged linear elastic stage and a subsequent softening damage process.

In the first stage, represented by the line with a positive slope, the cohesive traction  $\mathbf{t}_d$  versus relative displacement relation  $\boldsymbol{\delta}_r$  relationship can be described by

$$\mathbf{t}_d = \mathbb{K}_d \boldsymbol{\delta}_r = \begin{bmatrix} K_I & 0 & 0 \\ 0 & K_{II} & 0 \\ 0 & 0 & K_{III} \end{bmatrix} \boldsymbol{\delta}_r, \quad (3.51)$$

where  $\mathbb{K}_d$  is a stiffness fourth order tensor and  $K_I$ ,  $K_{II}$  and  $K_{III}$  are the normal and shear interface stiffnesses. They can also be named penalty stiffnesses and should have high values to avoid undesirable interpenetrations. However, from a certain value, numerical instabilities arise, meaning that a compromise is always necessary to select an appropriate value.

Even though the strain energy release rate and the penalty stiffness are mentioned like this, they are in fact parameters per unit of area and their units are N/mm and N/mm<sup>3</sup>, respectively.

The second stage is reached (and the damage initiates) when the relative displacement reaches the onset relative displacement,  $\delta_{o,i}$ . The degradation stage is what distinguishes each model since it is formulated differently for each one. In the simulation, failure occurs gradually mimicking what happens in the FPZ. Numerically, a damage parameter  $d_i$  is implemented who varies between zero (undamaged) and one (complete loss of stiffness).

For pure mode loading  $i$ , the damage parameter  $d_i$  is defined considering loading component on  $i$  direction (I, II or III). Anyway, stiffness properties of the three modes are degraded according to the softening relationship

$$\mathbf{t}_d = [\mathbf{I} - \mathbf{D}] \mathbb{K}_d \boldsymbol{\delta}_r = \begin{bmatrix} 1 - d_I & 0 & 0 \\ 0 & 1 - d_{II} & 0 \\ 0 & 0 & 1 - d_{III} \end{bmatrix} \mathbb{K}_d \boldsymbol{\delta}_r, \quad (3.52)$$

where  $\mathbf{D}$  is a diagonal matrix containing the damage parameters  $d_i$ .

The damage parameter from the triangular law is

$$d_i = \frac{\delta_{u,i}(\delta_i - \delta_{o,i})}{\delta_i(\delta_{u,i} - \delta_{o,i})}. \quad (3.53)$$

In it,  $\delta_i$  stands for the current relative displacement of the loading mode  $i$ ,  $\delta_{o,i}$  and  $\delta_{u,i}$  for the onset and the ultimate relative displacements of the softening stage.

Besides, it is also important to refer that the damage process is irreversible and so

$$d_i = \max(d_i^j, d_i^{j-1}). \quad (3.54)$$

The superscript  $j$  refers to the number of the increment in which the damage parameter is calculated.

So far, only pure mode loading situations are being considered. However, different loading modes (Figure 3.6) usually coexist and it is very usual for mixed-mode loadings to occur. Therefore, the previous model can be extended. Figure 3.5 also represents the mixed-mode model. The governing laws are very similar to the pure mode ones, but equivalent parameters (identified by a subscript  $m$ ) are used to define the damage parameter under mixed-mode loading. The area below this new model is the mode  $i$  component of the strain energy release rate ( $G_i$ ) under mixed-mode loading.

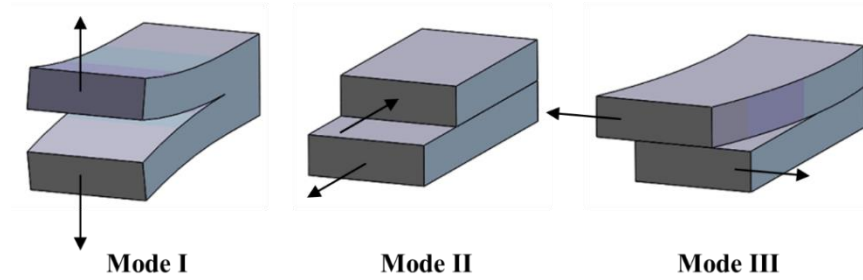


**Figure 3.6** – Failure modes. Adapted from Kashtalyan (2010).

In mixed-mode loading, damage initiation is determined by a quadratic stress criterion

$$\left(\frac{\sigma_I}{\sigma_{u,I}}\right)^2 + \left(\frac{\sigma_{II}}{\sigma_{u,II}}\right)^2 + \left(\frac{\sigma_{III}}{\sigma_{u,III}}\right)^2 = 1 \quad \text{if } \sigma_I \geq 0, \quad (3.55)$$

$$\left(\frac{\sigma_{II}}{\sigma_{u,II}}\right)^2 + \left(\frac{\sigma_{III}}{\sigma_{u,III}}\right)^2 = 1 \quad \text{if } \sigma_I \leq 0. \quad (3.56)$$

where  $\sigma_{u,I}$ ,  $\sigma_{u,II}$  and  $\sigma_{u,III}$  are, respectively, the mode I, II and III local strengths. It is assumed that normal compressive stresses do not promote damage – Equation (3.56).

Once a crack has initiated the above stress-based criterion cannot be used because of the stress singularity problems. Therefore, to simulate the mixed-mode damage propagation a fracture mechanics energetic criterion is used. A power law energetic one is chosen, and it is

$$\left\{\frac{G_I}{G_{IC}}\right\}^\varphi + \left\{\frac{G_{II}}{G_{IIC}}\right\}^\varphi + \left\{\frac{G_{III}}{G_{IIIC}}\right\}^\varphi = 1. \quad (3.57)$$

$G_{Ic}$ ,  $G_{IIc}$  and  $G_{IIIc}$  are the critical values in mode I, II and III, respectively, of the fracture energy and  $\varphi$  is the power and is a unique material parameter.

### 3.3. CONTACT MECHANICS

In a first instance, contact interactions can be mathematically interpreted as a group of nonlinear boundary conditions. However, neither the contact forces nor the displacements on the contact boundary are prescribed. In this section, important notions such as contact kinematics and contact conditions formulations are presented. General contact formulations accompanied by a comprehensive theoretical background can be found in Johnson (1985); Laursen (2003) and Wriggers (2006).

A typical general problem formulation where the problem setup considers contact interactions between two deformable bodies is assumed – Figure 3.7.

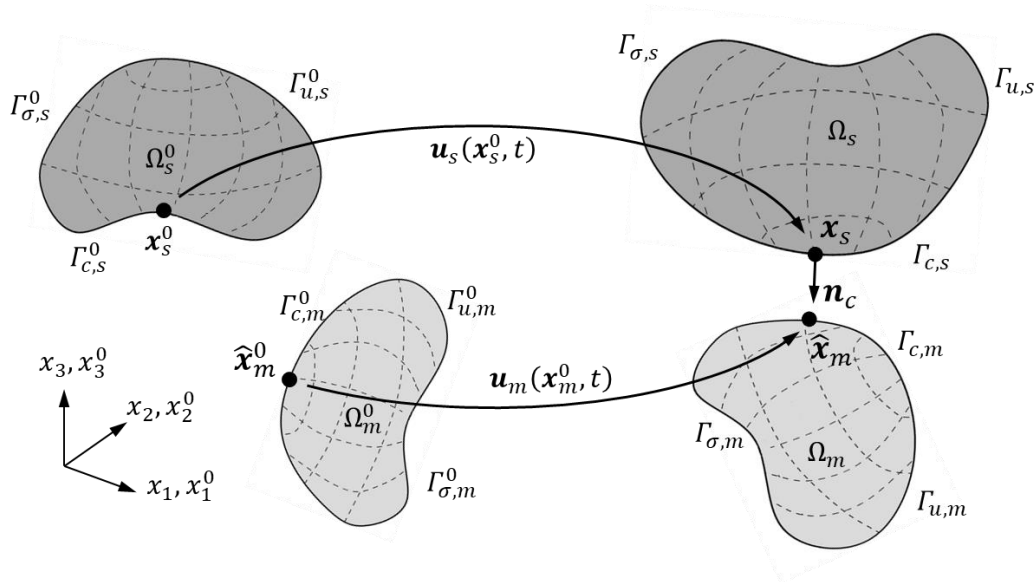


**Figure 3.7** – Contact between two bodies and respective nomenclature at the reference and deformed configurations. Adapted from Wriggers (2006).

Starting by extending the previously described problem formulation of nonlinear solid mechanics,  $\Omega_s^0, \Omega_m^0 \in \mathbb{R}^3$  and  $\Omega_s, \Omega_m \in \mathbb{R}^3$  are two (deformable) bodies in the initial and current configurations, respectively.

Focusing on the initial configuration, their boundaries  $\partial\Omega_i^0$ ,  $i = m, s$ , are divided into three disjoint subsets

$$\partial\Omega_i^0 = \Gamma_{u,i}^0 + \Gamma_{\sigma,i}^0 + \Gamma_{c,i}^0, \quad (3.58)$$

where  $\Gamma_{u,i}^0$  stands for the Dirichlet boundary where the displacements  $\bar{\mathbf{u}}$  are imposed,  $\Gamma_{\sigma,i}^0$  for the Neumann boundary where the tractions  $\bar{\mathbf{t}}$  are applied and  $\Gamma_{c,i}^0$  for the potential contact surface. Even though the master-slave concept should only be used in the context of FE discretisation, the subscripts of the previous surfaces refer to the master ( $m$ ) and slave ( $s$ ) surfaces.

Similarly, the boundaries of the current configuration  $\partial\Omega_i$ ,  $i = m, s$ , can be divided, but using the notation  $\Gamma_{u,i}$ ,  $\Gamma_{\sigma,i}$  and  $\Gamma_{c,i}$ .

It is important to highlight that a characteristic of contact problems is that the active contact surface  $\Gamma_{a,i} \in \Gamma_{c,i}$  is unknown and may change over time and thus must be determined as a part of the solution process.

To measure the proximity, potential contact, and penetration of the two bodies, a geometric measure named gap function is introduced in the current configuration. It can be defined as

$$g_n(\mathbf{x}_s, t) = -\mathbf{n}_c[\mathbf{x}_s - \hat{\mathbf{x}}_m((\mathbf{x}_s, t))]. \quad (3.59)$$

In it,  $\hat{\mathbf{x}}_m$  is the contact point on the master surface associated with each point  $\mathbf{x}_s$  of the slave surface and  $\mathbf{n}_c$  is its contact normal vector. The identification of the contact point on the master surface may be performed by a closest point projection

$$\hat{\mathbf{x}}_m = \arg \min_{\mathbf{x}_m \in \Gamma_{c,m}} \|\mathbf{x}_s - \mathbf{x}_m\|. \quad (3.60)$$

The  $\arg \min f(x)$  function provides the input  $x$  that results in the minimum of  $f(x)$ . So, in this context, the previous equation returns the point  $\mathbf{x}_m$ , from the potential contact master surface, closest to point  $\mathbf{x}_s$  of the slave surface.

Neglecting any tangential interactions in the contact (frictionless contact), the contact traction on the slave surface  $\mathbf{t}_{c,s}$  can be written by

$$\mathbf{t}_{c,s} = p_n \mathbf{n}_c, \quad (3.61)$$

where  $p_n$  is the normal contact pressure in the current configuration. Due to the equilibrium of linear momentum on the contact interface, the contact traction on the master surface is  $\mathbf{t}_{c,m} = -\mathbf{t}_{c,s}$ .

For the normal contact constraints, the KKT conditions are used

$$g_n(\mathbf{x}^0, t) \geq 0, \quad (3.62)$$

$$p_n(\mathbf{x}^0, t) \leq 0, \quad (3.63)$$

$$p_n(\mathbf{x}^0, t)g_n(\mathbf{x}^0, t) = 0. \quad (3.64)$$

These define a non-adhesive and frictionless contact. Equation (3.62) represents the geometric constraint of non-penetration, Equation (3.63) implies that no adhesive stresses are allowed in the contact zone and Equation (3.64) forces the gap to be closed when contact occurs (non-zero contact pressure) and the contact pressure to be zero when there is no contact (gap is open) – Figure 3.8.

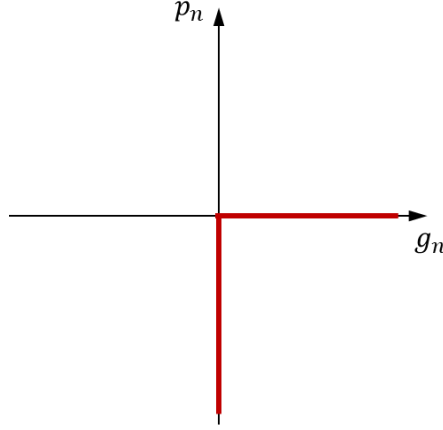


**Figure 3.8** – Graphical representation of the KKT contact conditions. Adapted from Laursen (2003).

In the context of contact dynamics, an additional contact condition called the persistency condition must be considered

$$p_n(\mathbf{x}^0, t) \dot{g}_n(\mathbf{x}^0, t) = 0, \quad (3.65)$$

where  $\dot{g}_n(\mathbf{x}^0, t)$  is the material time derivative of the gap function. This condition assures that the contact pressure is only non-zero when the bodies are in contact.

### 3.4. WEAK FORMULATION

The last formulation step in the continuum framework is the derivation of the weak form equations. These are the point of departure for the derivation of FE approximations.

The functional space of the displacements and virtual displacements (also commonly termed solution and weighting spaces) are denoted  $\mathbf{u}_i$  and  $\mathbf{v}_i$ , respectively, and can be defined as

$$\mathbf{u}_i = \{\mathbf{u}_i \in H^1(\Omega_i) \mid \mathbf{u}_i = \bar{\mathbf{u}}_i \text{ on } \Gamma_{u,i}\}, \quad (3.66)$$

$$\mathbf{v}_i = \{\delta \mathbf{u}_i \in H^1(\Omega_i) \mid \delta \mathbf{u}_i = 0 \text{ on } \Gamma_{u,i}\}. \quad (3.67)$$

$\delta \mathbf{u}_i$  is the virtual displacement field and  $H^1(\Omega_i)$  is the Sobolev functional space over  $\Omega$ , which contains all functions with square-integrable up to first-order derivatives in the current configuration domain.

The Principle of Virtual Work (PVW) states that the virtual work produced by a system of forces at equilibrium subjected to any cinematically admissible virtual displacement field is zero. The purpose of its application is to find  $\mathbf{u} \in \mathbf{u}$  such that for all admissible  $\delta \mathbf{u} \in \mathbf{v}$  fulfill

$$\delta \Pi(\mathbf{u}, \delta \mathbf{u}) = \delta \Pi_{kin}(\mathbf{u}, \delta \mathbf{u}) + \delta \Pi_{int}(\mathbf{u}, \delta \mathbf{u}) + \delta \Pi_{ext}(\mathbf{u}, \delta \mathbf{u}) + \delta \Pi_c(\mathbf{u}, \delta \mathbf{u}) = 0. \quad (3.68)$$

The multiple contributions to the virtual work of the system  $\delta\Pi$  are detailed next: the contribution of the kinetic or inertial virtual work  $\delta\Pi_{kin}$  (necessary in dynamic problems) is

$$\delta\Pi_{kin}(\mathbf{u}, \delta\mathbf{u}) = \sum_{i \in \{s, m\}} \delta\Pi_{kin,i}(\mathbf{u}_i, \delta\mathbf{u}_i) = - \sum_{i \in \{s, m\}} \int_{\Omega_i} \rho_i \ddot{\mathbf{u}}_i \cdot \delta\mathbf{u}_i \, d\Omega ; \quad (3.69)$$

the virtual work of the internal forces  $\delta\Pi_{int}$  that includes the internal force of the bulk material and the internal cohesive force and is

$$\begin{aligned} \delta\Pi_{int}(\mathbf{u}, \delta\mathbf{u}) = \sum_{i \in \{s, m\}} \delta\Pi_{int,i}(\mathbf{u}_i, \delta\mathbf{u}_i) = & - \sum_{i \in \{s, m\}} \int_{\Omega_i} \mathbf{S} : \delta\mathbf{E} \, d\Omega \\ & - \sum_{i \in \{s, m\}} \int_{\Gamma_{d,i}} \mathbf{t}_d \cdot \delta\boldsymbol{\delta}_{r,i} \, d\Gamma ; \end{aligned} \quad (3.70)$$

the virtual work of the external loads  $\delta\Pi_{ext}$  considers the contribution of the body forces and surface tractions and is

$$\begin{aligned} \delta\Pi_{ext}(\mathbf{u}, \delta\mathbf{u}) = \sum_{i \in \{s, m\}} \delta\Pi_{ext,i}(\mathbf{u}_i, \delta\mathbf{u}_i) = & \sum_{i \in \{s, m\}} \int_{\Omega_i} \mathbf{b}_i \cdot \delta\mathbf{u}_i \, d\Omega \\ & + \sum_{i \in \{s, m\}} \int_{\Gamma_{\sigma,i}} \mathbf{t}_i \cdot \delta\mathbf{u}_i \, d\Gamma ; \end{aligned} \quad (3.71)$$

the virtual work related with the contact interactions  $\delta\Pi_c$  is

$$\delta\Pi_c(\mathbf{u}, \delta\mathbf{u}) = \sum_{i \in \{s, m\}} \delta\Pi_{c,i}(\mathbf{u}_i, \delta\mathbf{u}_i) = \sum_{i \in \{s, m\}} \int_{\Gamma_{c,i}} \mathbf{t}_{c,i} \cdot \delta\mathbf{u}_i \, d\Gamma . \quad (3.72)$$

Since there is a time dependency, initial conditions specifying the displacements  $\bar{\mathbf{u}}_i^0(\mathbf{x})$  and velocities  $\bar{\dot{\mathbf{u}}}_i^0(\mathbf{x})$  at  $t = 0$  need to be verified

$$\mathbf{u}_i(\mathbf{x}, 0) = \bar{\mathbf{u}}_i^0(\mathbf{x}) \text{ in } \Omega_i^0 , \quad (3.73)$$

$$\dot{\mathbf{u}}_i(\mathbf{x}, 0) = \bar{\dot{\mathbf{u}}}_i^0(\mathbf{x}) \text{ in } \Omega_i^0 . \quad (3.74)$$

All the above equations do not assume any simplification or constitutive law and so are valid and applicable for any of the mentioned problems in section 3.2.5. The hyperelasticity, the elastoplasticity and the damage degradation are accounted for in the first term of the internal virtual work.

In the present model, the constraints are enforced using the Lagrange multiplier method. This method has the downside of adding new unknowns, which increases the size of the system of equations. However, it allows us to satisfy exactly the constraints. The weak normal constraints, for all admissible  $\delta\boldsymbol{\lambda} \in \mathcal{M}$ , are given as follows

$$\int_{\Gamma_{c,s}} g_n(\delta\boldsymbol{\lambda} - \boldsymbol{\lambda}) \, d\Gamma \geq 0 . \quad (3.75)$$

$\lambda$  is the Lagrange multiplier vector and  $\mathcal{M}$  represents the dual space of the trace space  $\mathcal{W}_s$  of  $\mathcal{V}_s$ , where each of its scalar components reads  $\mathcal{W} = H^{1/2}(\Gamma_{c,s})$ . The components of the solution space  $\mathcal{M}$  are  $\mathcal{M} = H^{-1/2}(\Gamma_{c,s})$ .

### 3.5. FINITE ELEMENT METHOD

This chapter is dedicated to the derivation of the discrete version of the weak form following from the application of the FEM. Initially the domain is discretised spatially and next in the time variable. For more details on FE spatial discretisations, the textbooks Zienkiewicz and Taylor (2000b, 2000a) and Popp (2012) are referred to and for time discretisation, the Hilber et al. (1977) and Bathe (2014) ones.

#### 3.5.1. SPACE DISCRETISATION

To apply the  $h$ -version FEM, the continuum  $\Omega^0$  must be divided into  $n_e$  smaller domains  $\Omega_e^0$

$$\Omega^0 \approx \Omega_h^0 = \bigcup_{e=1}^{n_e} \Omega_e^0. \quad (3.76)$$

$\Omega_h^0$  denotes a FE approximation to the continuum domain  $\Omega^0$ . All variables with the subscript  $h$  refer to the FE approximation of some quantity.

The displacement solution  $\mathbf{u}_e$  is typically approximated by local interpolation functions  $N_k$  named shape functions

$$\mathbf{u}_e(\mathbf{x}^0, t) \approx \mathbf{u}_{e,h}(\mathbf{x}^0, t) = \sum_{k=1}^{n_{nod}} N_k(\mathbf{x}^0) \mathbf{d}_k(t), \quad (3.77)$$

where  $n_{nod}$  denotes the number of nodes associated with each element  $e$  and  $\mathbf{d}_k$  is the nodal displacement vector.

Considering an analysis based on isoparametric elements, the functions used to interpolate the element's geometry  $\mathbf{x}_{e,h}^0$  or  $\mathbf{x}_{e,h}$  and its displacement field  $\mathbf{u}_{e,h}$  are the same. Also, based on the isoparametric concept,  $\Omega_e^0$  may be mapped to a reference element geometry.

Thus, the interpolation of displacements, current geometry and reference geometry at the element are expressed as

$$\mathbf{u}_{e,h}(\xi, t) = \sum_{k=1}^m N_k(\xi) \mathbf{d}_k(t), \quad (3.78)$$

$$\mathbf{x}_{e,h}(\xi, t) = \sum_{k=1}^m N_k(\xi) \mathbf{x}_k(t), \quad (3.79)$$

$$\mathbf{x}_{e,h}^0(\xi, t) = \sum_{k=1}^m N_k(\xi) \mathbf{x}_k^0, \quad (3.80)$$

where  $\mathbf{x}_k(t)$  and  $\mathbf{x}_k^0$  are the vectors of nodal positions in the current and reference configurations.

Using the previous interpolations into the weak formulation, the final discretised formulation, after integration, emerges as

$$\mathbf{M}\ddot{\mathbf{d}} + \mathbf{F}_{int}(\mathbf{d}) = \mathbf{F}_{ext}(t) - \mathbf{F}_c(\mathbf{d}, \lambda). \quad (3.81)$$

$\mathbf{M}$  is the global mass matrix and it contains the contribution of kinetic forces,  $\mathbf{F}_{int}$  is the global vector of internal forces,  $\mathbf{F}_{ext}$  is the global vector of external forces and  $\mathbf{F}_c$  is the global vector of the contact forces. All of them are obtained by assembling the contribution from each elemental structure.  $\ddot{\mathbf{d}}$  and  $\mathbf{d}$  are global vectors of all discrete nodal values of accelerations and displacements, respectively.

To take energy dissipation into account a global damping matrix  $\mathbf{C}$  can be introduced in Equation (3.81)

$$\mathbf{M}\ddot{\mathbf{d}} + \mathbf{C}\dot{\mathbf{d}} + \mathbf{F}_{int}(\mathbf{d}) = \mathbf{F}_{ext}(t) - \mathbf{F}_c(\mathbf{d}, \lambda). \quad (3.82)$$

$\dot{\mathbf{d}}$  is the global vector of all discrete nodal values of velocities. Such as the other matrices, the global damping matrix can be obtained by assembling the contribution from each elemental damping matrix  $\mathbf{C}_e$ . Using Rayleigh's damping, these are a linear combination of the elemental mass matrix  $\mathbf{M}_e$  and the elemental initial tangent stiffness matrix for each element  $\mathbf{K}_e^{init}$

$$\mathbf{C}_e = \alpha_e \mathbf{M}_e + \beta_e \mathbf{K}_e^{init}, \quad (3.83)$$

where  $\alpha_e$  and  $\beta_e$  are proportional damping constants of each element.

Equation (3.82) traduces a system of  $n_{nod} \times n_e$  nonlinear ordinary differential equations. These must be subjected to the initial conditions of Equations (3.73) and (3.74), to the boundary conditions on  $\Gamma_{u,h}$  and impenetrability conditions from Equations (3.62) to (3.64) on  $\Gamma_{c,h}$ .

The discretisation of the contact interface is accomplished by the contact algorithm. In computational contact mechanics, the contact algorithm must be responsible for the discretisation of the virtual work and for the enforcement of the contact constraints. In this project, a surface-to-surface contact algorithm is applied. More about this matter can be found in Smith (2009f).

Compared to node-to-surface discretisation, surface-to-surface discretisation provides more accurate stress and pressure results and is less sensitive to master and slave surface designations. However, it has a bigger computational cost.

Surface-to-surface discretisation considers the shape of both the slave and master surfaces in the region of the contact constraints and has the following key characteristics:

- The surface-to-surface formulation enforces contact conditions in an average sense over regions nearby slave nodes rather than only at individual slave nodes like in node-to-surface contact. The averaging regions are approximately centred on slave nodes, so each contact constraint will predominantly consider one slave node but will also consider adjacent slave nodes. Some penetration may be observed at individual nodes, however, large, undetected penetrations of master nodes into the slave surface do not occur with this discretisation;

- The contact direction is based on an average normal of the slave surface in the region surrounding a slave node.

The active contact surface is calculated using a finite-sliding tracking approach. This allows for arbitrary separation, sliding, and rotation of the surfaces and it is recommended for problems with large deformations or rotations (Smith, 2009f).

### 3.5.2. TIME DISCRETISATION

The semi-discrete system of Equation (3.82) must be completely discretised by applying an approximation in the time domain.

Time integration in structural mechanics can be applied after the equations of motion are transformed to a more convenient set of coordinates, usually using the mode shapes of vibration (mode superposition), or directly applied to the original system of equations of motion. In linear systems, the mode superposition method has the advantage of obtaining a decoupled set of ordinary differential equations. However, the second approach is chosen, because even though it has a bigger computational cost, it allows dealing with nonlinear problems more easily. To this end, the time domain is discretised in a set of intervals  $\Delta t$  where  $t \in [t_n, t_{n+1}]$  and the approximate solution is found at the end of each interval.

Direct integration methods can be further divided into explicit and implicit. Explicit schemes are based on the verification of the temporally discrete equations on time step  $t_n$  to compute the response on  $t_{n+1}$ . This methodology has the advantage of requiring very short computation times per time step but is conditionally stable and thus many very small-time steps must be used. Implicit schemes verify the discrete equilibrium equations at time  $t_{n+1}$  in order to determine the displacement at the same time step. Its major perk is, with an appropriate choice of parameters, being unconditionally stable in the linear regime, so even though each time step is more expensive, it allows for larger, and fewer time steps to be employed.

In this study, the HHT temporal integrator developed by Hilber et al. (1977) is considered. This scheme is a second order accurate, unconditionally stable with controllable numerical damping scheme. The  $\alpha$  -dissipation is introduced in the integration by shifting the evaluation of the internal, damping and contact forces to a generalised midpoint  $t_{n+\alpha_{HHT}}$ , with  $\alpha_{HHT} \leq 0$ , so that Equation (3.82) is replaced by

$$\mathbf{M}\ddot{\mathbf{d}}_{n+1} + \mathbf{C}\dot{\mathbf{d}}_{n+\alpha_{HHT}} + \mathbf{F}_{int}(\mathbf{d}_{n+\alpha_{HHT}}) = \mathbf{F}_{ext,n+\alpha_{HHT}} - \mathbf{F}_c(\mathbf{d}_{n+\alpha_{HHT}}, \boldsymbol{\lambda}_{n+\alpha_{HHT}}), \quad (3.84)$$

with the initial conditions  $\mathbf{d}_0 = \hat{\mathbf{d}}$  and  $\dot{\mathbf{d}}_0 = \hat{\dot{\mathbf{d}}}$ , given displacement boundary conditions and boundary conditions being calculated due to developing contact.

The nodal quantities associated the generalised midpoints are given by linear combinations of the associated endpoint variables

$$\mathbf{d}_{n+\alpha_{HHT}} = \alpha_{HHT} \mathbf{d}_{n+1} + (1 - \alpha_{HHT}) \mathbf{d}_n, \quad (3.85)$$

$$\dot{\mathbf{d}}_{n+\alpha_{HHT}} = \alpha_{HHT} \dot{\mathbf{d}}_{n+1} + (1 - \alpha_{HHT}) \dot{\mathbf{d}}_n, \quad (3.86)$$

$$\mathbf{F}_{ext,n+\alpha_{HHT}} = \alpha_{HHT} \mathbf{F}_{ext,n+1} + (1 - \alpha_{HHT}) \mathbf{F}_{ext,n}, \quad (3.87)$$

where the displacements and velocities are given by the Newmark interpolation formulas

$$\mathbf{d}_{n+1} = \mathbf{d}_n + \Delta t \dot{\mathbf{d}}_n + \frac{\Delta t^2}{2} [(1 - 2\beta_N) \ddot{\mathbf{d}}_n + 2\beta_N \ddot{\mathbf{d}}_{n+1}], \quad (3.88)$$

$$\dot{\mathbf{d}}_{n+1} = \dot{\mathbf{d}}_n + \Delta t [(1 - \gamma_N) \ddot{\mathbf{d}}_n + \gamma_N \ddot{\mathbf{d}}_{n+1}], \quad (3.89)$$

and, for each  $\alpha_{HHT}$ , the  $\beta_N$  and  $\gamma_N$  coefficients that ensure second-order accuracy and unconditional stability are

$$-\frac{1}{3} \leq \alpha_{HHT} \leq 0, \quad \beta_N = \frac{1}{4} (1 - \alpha_{HHT})^2, \quad \gamma_N = \frac{1}{2} - \alpha_{HHT}. \quad (3.90)$$

The time integrator leads to a nonlinear algebraic problem which must be solved at each time step. For this, the Newton–Raphson iterative method is selected due to its quadratic convergence rate. Hence, the linearized problem should be solved for each iteration  $j$  as

$$\begin{aligned} & \frac{\partial}{\partial \mathbf{d}} [\mathbf{M} \ddot{\mathbf{d}} + \mathbf{C} \dot{\mathbf{d}} + \mathbf{F}_{int}(\mathbf{d}) + \mathbf{F}_c(\mathbf{d}, \boldsymbol{\lambda})] \Big|_{\mathbf{d}_{n+\alpha_{HHT}}^j} \Delta \mathbf{d} \\ & = \mathbf{F}_{ext,n+\alpha_{HHT}} - \mathbf{M} \ddot{\mathbf{d}}_{n+1}^j - \mathbf{C} \dot{\mathbf{d}}_{n+\alpha_{HHT}}^j - \mathbf{F}_{int}(\mathbf{d}_{n+\alpha_{HHT}}^j) - \mathbf{F}_c(\mathbf{d}_{n+\alpha_{HHT}}^j, \boldsymbol{\lambda}_{n+\alpha_{HHT}}^j), \end{aligned} \quad (3.91)$$

with iterations updated by

$$\mathbf{d}_{n+1}^{j+1} = \mathbf{d}_{n+1}^j + \Delta \mathbf{d}. \quad (3.92)$$

### 3.6. FINAL REMARKS

During this chapter, the mathematical model on which the following analyses are based is introduced. Note that:

- the model is built from nonlinear continuum and contact mechanics concepts;
- besides accounting for the elastic behaviour of materials, it also considers elastoplastic and damaged materials;
- to achieve the full discrete version of the weak form, the space discretisation is performed in agreement with the  $h$ -version FEM and the time discretisation is made by a HHT temporal integrator.

The pertinence of these model's characteristics is evident in Chapter 5, Chapter 6 and Chapter 7. All aspects of the discussed model are already implemented in the ABAQUS/Standard code, used throughout this thesis, so, such features need only to be activated.



# CHAPTER 4. WING REPRESENTATIVE STRUCTURE'S DESCRIPTION

---

## 4.1. INTRODUCTION

Throughout this chapter, the wing representative structure is explored. chapter 4.2 presents the structure's geometry, constitution, and materials. Next, all mechanical properties needed to develop this thesis are listed according to the division made in section 3.2.5, with the main purpose of materialising the previously presented constitutive models with numerical values.

In chapter 4.3, in addition to consulting the available literature, micromechanics-based models are used to deduce a composite's mechanical properties by extrapolating from existing data for a similar composite. In chapters 4.4 and 4.5, elastoplastic and damaged material properties are discussed.

## 4.2. WING REPRESENTATIVE STRUCTURE'S GEOMETRY

First of all, the components of the structure under analysis are represented in Figure 4.1. The connection between components is assured by bolts in most of the scenarios. Besides the bolts, Araldite® 420 A/B is used between the stiffeners and the panels.

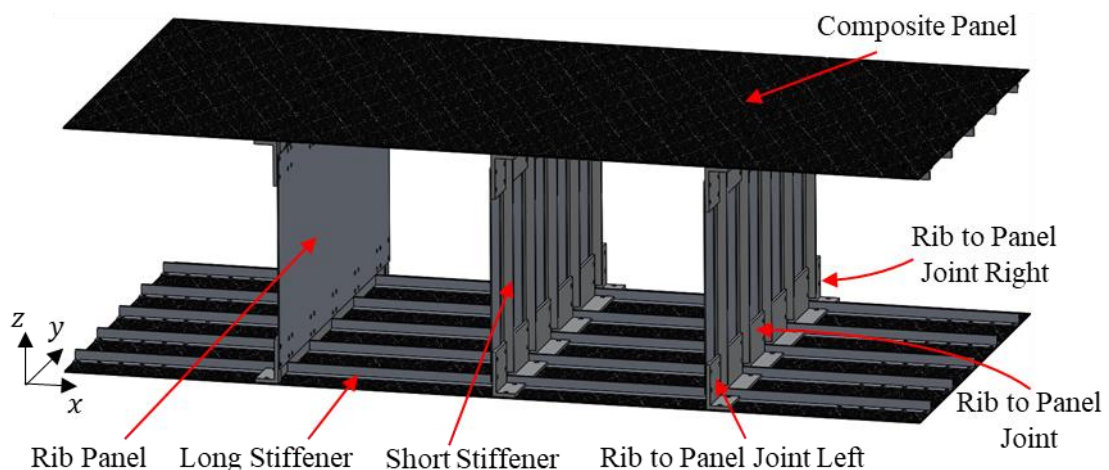


**Figure 4.1** – Wing representative structure and its components. Drawing by School of Engineering, Cardiff University.

When performing the static analysis, a test structure is added to function like a rigid body and transmit the test efforts. Figure 4.2 presents its components.

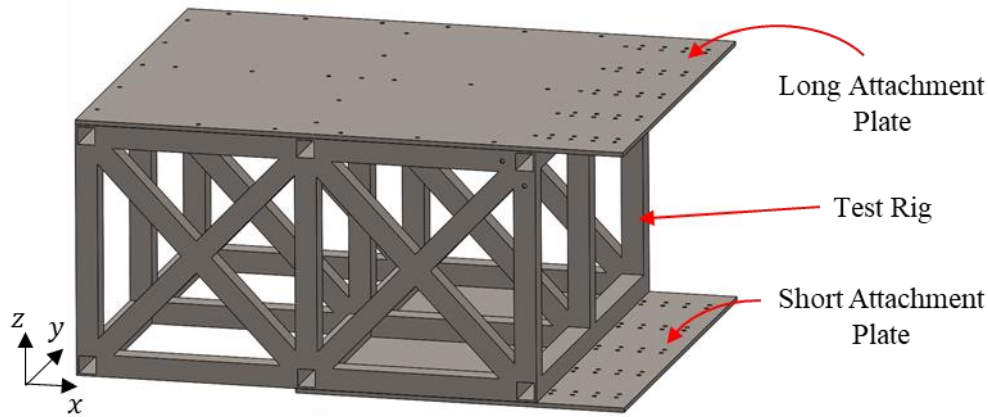


**Figure 4.2** – Test structure and its components. Drawing by School of Engineering, Cardiff University.

The quantities needed for the wing representative structure and the testing one and the materials used are listed in Table 4.1.

**Table 4.1** – List of the structure's constituents and their materials.

| Part Designation              | Quantity | Material                                                       |
|-------------------------------|----------|----------------------------------------------------------------|
| Composite panel               | 2        | 18 plies of USN150B<br>(-45,0,45,0,90,0,-45,0,45) <sub>s</sub> |
| Long stiffener                | 10       | Aluminium 6063AT6                                              |
| Short stiffener               | 15       | Aluminium 6063AT6                                              |
| Rib panel                     | 3        | Aluminium 6082AT6                                              |
| Rib to panel joint            | 24       | Aluminium 6082AT6                                              |
| Rib to panel joint half left  | 6        | Aluminium 6082AT6                                              |
| Rib to panel joint half right | 6        | Aluminium 6082AT6                                              |
| Long attachment plate         | 2        | Steel BS EN10025 S275JR                                        |
| Short attachment plate        | 2        | Steel BS EN10025 S275JR                                        |
| Rig                           | 2        | Steel EN10219 S235JRH                                          |

Depending on the analysis and on the material, different material behaviour can be considered. Initially, all the materials are modelled as elastic ones, which is the scenario for the steel alloys and the 6082AT6 aluminium

However, since the modification of material properties with the development of the analysis is of interest during this thesis, the plastic behaviour of the aluminium 6063AT6 is discussed in section 4.4 and the damaged response of the USN150B composite and Araldite® 420 A/B is shown in section 4.5.

### 4.3. ELASTIC MATERIALS

The elastic properties and density of aluminium and steel alloys and of Araldite® 420 A/B used throughout this report are summarised in Table 4.2. The properties of aluminium are from ANSYS Granta © (2020), the ones of steel are from The British Standards Institution (2006, 2019) and the ones of Araldite® 420 A/B are from de Moura et al. (2006) and Huntsman (2012). All these materials are considered isotropic.

**Table 4.2** – Elastic properties of aluminium and steel alloys and of Araldite® 420 A/B.

| Material                | $\sigma_y$ [MPa] | $E$ [MPa] | $\nu$ | $\rho$ [g. cm <sup>-3</sup> ] |
|-------------------------|------------------|-----------|-------|-------------------------------|
| Aluminium 6063AT6       | 170              | 70000     | 0.33  | 2.70                          |
| Aluminium 6082AT6       | 260              |           |       |                               |
| Steel EN10219 S235JRH   | 235              | 210000    | 0.3   | 7.5                           |
| Steel BS EN10025 S275JR | 275              |           |       |                               |
| Araldite® 420 A/B       | 27               | 1850      | 0.3   | 1.2                           |

The composite panel is constituted by 18 stacked plies of USN150B (transversely isotropic carbon fibre-epoxy material), whose properties are not fully supplied by the manufacturer. However, a very similar composite named USN150A has a much more detailed datasheet. Therefore, knowing that the USN150A and the USN150B composites are formed by the same SKYFLEX K51 matrix and TR50S15L fibres and only their fibre content changes, the mechanical properties of USN150B are extrapolated using micromechanics-based models.

To do so, in chapter 4.3.1 the fibres and matrixes of both composites are characterised and in chapter 4.3.2 the properties of USN150B are deduced and compared with the existing literature.

#### 4.3.1. FIBRE AND MATRIX CHARACTERISATION

In Table 4.3, the already known properties of TR50S15L fibres and SKYFLEX K51 resin, from Mitsubishi Chemical (2020) and SK Chemicals (2013), respectively, are listed and, in Table 4.4, the USN150A's composition and some of its mechanical properties are presented (SK Chemicals, 2013). The subscript  $f$  refers to parameters of the fibres,  $R$  to ones of the resin/matrix and  $A$  to the ones of USN150A.  $RC$  and  $FC$  are the resin and fibre contents.

**Table 4.3** – Elastic properties of TR50S15L fibres and SKYFLEX K51 resin.

| $\sigma_{y,f}$ [GPa] | $E_{11,f}$ [GPa] | $\rho_f$ [g. cm <sup>-3</sup> ] | $\rho_R$ [g. cm <sup>-3</sup> ] |
|----------------------|------------------|---------------------------------|---------------------------------|
| 4.9                  | 235              | 1.82                            | 1.20                            |

**Table 4.4** – USN150A's composition and elastic properties.

| $RC_A$ [%] | $FC_A$ [%] | $E_{11,A}$ [GPa] | $E_{22,A}$ [GPa] |
|------------|------------|------------------|------------------|
| 36.0       | 64.0       | 125.3            | 8.4              |

For any composite, disregarding any voids, the resin and fibre contents can be written as

$$RC = \frac{m_R}{m_R + m_f}, \quad (4.1)$$

$$FC = \frac{m_f}{m_R + m_f}, \quad (4.2)$$

where  $m_R$  and  $m_f$  are, respectively, the resin and fibre masses.

Using the concept of total weight  $m_T$ , which is the sum of the previously mentioned masses, it is possible to further develop the former equations and write the resin and fibre masses as

$$m_R = RC \cdot (m_R + m_f) = RC \cdot m_T, \quad (4.3)$$

$$m_f = FC \cdot (m_R + m_f) = FC \cdot m_T. \quad (4.4)$$

Knowing this, the composite's density  $\rho_c$  and its fibre volume fraction  $V_f$  can be described as a function of the mass contents:

$$\rho_c = \frac{m_T}{Vol_T} = \frac{m_T}{Vol_R + Vol_f} = \frac{m_T}{\frac{m_T \cdot RC}{\rho_R} + \frac{m_T \cdot FC}{\rho_f}} = \frac{1}{\frac{RC}{\rho_R} + \frac{FC}{\rho_f}}, \quad (4.5)$$

$$V_f = \frac{Vol_f}{Vol_R + Vol_f} = \frac{\frac{m_T \cdot FC}{\rho_f}}{\frac{m_T \cdot RC}{\rho_R} + \frac{m_T \cdot FC}{\rho_f}} = \frac{\frac{FC}{\rho_f}}{\frac{RC}{\rho_R} + \frac{FC}{\rho_f}} = \frac{FC}{\rho_f} \cdot \rho_c. \quad (4.6)$$

For USN150A, the obtained density is  $\rho_{c,A} = 1.535 \text{ g/cm}^3$  and its fibre volume fraction is  $V_{f,A} = 53.963 \%$ .

Using, now, micromechanics-based models on the USN150A's elastic properties, it is possible to estimate the matrix and fibre's missing properties. Next, the Voigt model (Voigt, 1889) of the Rule of Mixtures and the model for transverse modulus based on the unit cell (Fu et al., 1998) are used.

These models are developed from the following hypothesis (Raju et al., 2018):

- Composite materials have only two phases: fibres and matrix, which are bonded perfectly;
- Fibres are unidirectional and uniformly placed;
- Polymer matrix is considered isotropic;
- Composite material is considered transversely isotropic (properties are constant in the perpendicular plane to the fibre direction);
- Properties predicted are according to the principal material axes.

The Voigt model is a very simple model that assumes that under longitudinal loading conditions, the fibre's strain is the same as the matrix's strain. Using it, the effective longitudinal Young's modulus can be obtained by

$$E_{11} = E_{11,f}V_f + E_R(1 - V_f). \quad (4.7)$$

Rewriting the previous equation, the value calculated for the matrix's Young's modulus is  $E_R = -3.286$  GPa. Obviously, this negative value disrespects the physical meaning of Young's modulus. It may be due to the existence of unaccounted voids in the composite. Nonetheless, it is still used to estimate the elasticity moduli of composite USN150B, as if a proportionality rule was applied and, indirectly, taking void content into account.

In the model for transverse modulus based on the unit cell, the transverse Young's modulus of the composite is predicted by

$$\frac{1}{E_{22}} = \frac{\sqrt{\frac{4V_f}{\pi}}}{E_{22,f}\sqrt{\frac{\pi V_f}{4}} + E_m \left(1 - \sqrt{\frac{\pi V_f}{4}}\right)} + \frac{1 - \sqrt{\frac{4V_f}{\pi}}}{E_m}. \quad (4.8)$$

This expression is more reasonable than the rule of mixtures type expressions based on isostress or isostrain conditions because it results from the equilibrium of the diverse parts that constitute a volume element of the composite. The same criticism does not apply to Equation (4.7), which also results from an equilibrium of a volume element in the longitudinal direction.

Once again, rewriting the last equation, the value of the fibre's transverse Young's modulus can be estimated as  $E_{22,f} = 9.202$  GPa.

### 4.3.2. USN150B'S PROPERTIES

Employing now the equations presented in the previous section, the longitudinal and transverse Young's moduli of the USN150B composite can be predicted. As a starting point, Table 4.5 presents the USN150B's resin and fibre contents (SK Chemicals, 2013). From now on, the subscript  $B$  refers to parameters of USN150B.

**Table 4.5** – USN150B's composition.

| $RC_B$ [%] | $FC_B$ [%] |
|------------|------------|
| 33.0       | 67.0       |

With this information, the density and fibre volume fraction (Equations (4.5) and (4.6)) are  $\rho_{c,B} = 1.555$  g/cm<sup>3</sup> and  $V_{f,B} = 57.24$  %.

Also, the longitudinal and transverse Young's moduli of USN150B (Equations (4.7) and (4.8)) are  $E_{11,B} = 133.110$  GPa and  $E_{22,B} = 8.111$  GPa.

After searching in the available literature, Kim et al. (2015) and Shin and Lee (2003) presented the most similar properties to the ones predicted for USN150B. From these, the ones from Kim et al. (2015) were chosen, with Poisson's  $\nu_{12,B}$  ratio adapted according to

$$\nu_{12,B} = \frac{E_{11,B}\nu_{21,B}}{E_{22,B}} = \frac{131.0 \cdot 0.018}{8.0} = 0.29, \quad (4.9)$$

and are listed in Table 4.6.  $X$ ,  $Y$  and  $Z$  are the normal strengths in direction 1, 2 and 3, respectively, the subscript  $t$  and  $c$  refer to tensile and compressive, and  $S$  is the shear strength.

**Table 4.6** – USN150B's elastic properties.

|                                    |       |                             |          |
|------------------------------------|-------|-----------------------------|----------|
| $E_{11,B}$ [GPa]                   | 131.0 | $X_{t,B}$ [MPa]             | 2000.0   |
| $E_{22,B} = E_{33,B}$ [GPa]        | 8.0   | $Y_{t,B} = Z_{t,B}$ [MPa]   | 61.0     |
| $G_{12,B} = G_{13,B}$ [GPa]        | 4.5   | $X_{c,B}$ [MPa]             | 2000.0   |
| $G_{23,B}$ [GPa]                   | 3.5   | $Y_{c,B} = Z_{c,B}$ [MPa]   | 200.0    |
| $\nu_{12,B} = \nu_{13,B}$          | 0.29  | $S_{12,B} = S_{13,B}$ [MPa] | 70.0     |
| $\nu_{23,B}$                       | 0.47  | $S_{23,B}$ [MPa]            | 40.0     |
| $\rho_{c,B}$ [g·cm <sup>-3</sup> ] | 1.544 | $t_{ply,B}$ [mm]            | 0.194444 |

Additionally, in Appendix A the calculation of Young's modulus of the laminated panel is carried out using the classical laminated plate theory. The obtained value, 70.97 GPa, lies between the experimentally determined values of the compressive and tensile moduli of elasticity – respectively 65 GPa and 76 GPa – provided by Dr. Stephen Grigg, School of Engineering, Cardiff University within the COST action. Therefore, it is possible to conclude that the previous predictions are reasonable estimates.

#### 4.4. ELASTOPLASTIC MATERIALS

To apply the constitutive laws for elastoplastic materials described in section 3.2.5, it is only missing to fully define the hardening law used. Observing the stress-strain curves of Hilditch et al. (2009); Yang and Qi (2013) and Kohar et al. (2016), one can conclude about the behaviour of aluminium 6063AT6: just like suggested in the model formulation, a good approximation of its response is a linear isotropic hardening law. A scheme of this referred law is presented in Figure 4.3.

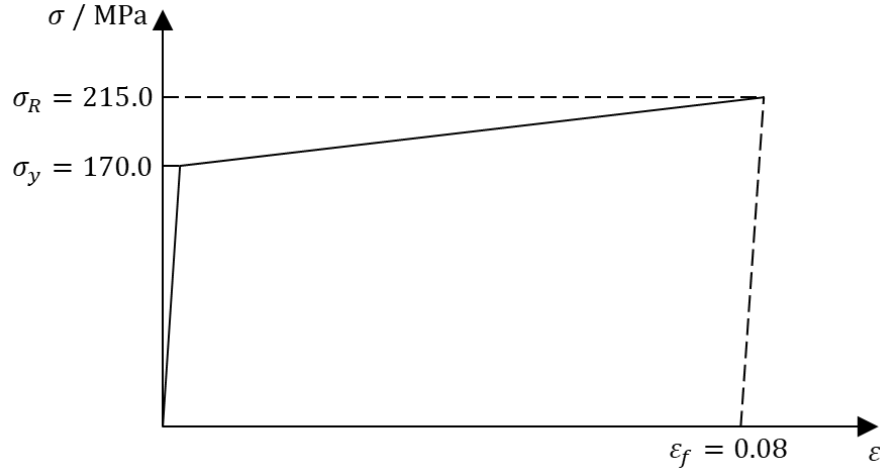


**Figure 4.3** – Stress-strain approximation for aluminium 6063AT6.

The first line represents the elastic stage and is characterised by a slope of value  $E = 70000$  MPa and a maximum value of  $\sigma_y = 170$  MPa. To characterise the second stage, the plastic one where the permanent deformation is acquired, it is important to state that the ultimate stress  $\sigma_R$  of aluminium 6063AT6 is 215 MPa and that the elongation at break  $\epsilon_f$  is 8%. With these values, it is possible to distinguish this stage by a maximum value of  $\sigma_R = 215$  MPa and a slope of

$$H = \frac{215.0 - 170.0}{0.08 - 0.0} = 562.5 \text{ MPa} . \quad (4.10)$$

## 4.5. DAMAGED MATERIALS

Like mentioned in section 3.2.5, a triangular traction-separation law is assumed – Figure 3.5. To fully characterise it, there is the need to quantify, for each mode  $i = I, II, III$ , the penalty stiffness  $K_i$ , the strength  $\sigma_{u,i}$  and the critical value of the fracture energy  $G_{ic}$ . Also, to establish the exact mixed-mode damage propagation criterion, the power  $\varphi$  must be quantified too.

Throughout this report, the damage mechanisms treated are the delamination of the composite laminate and the debonding in the Araldite® 420 A/B adhesive between the stiffeners and the same laminate.

According to Dourado et al. (2012), values like  $K = K_I = K_{II} = K_{III} = 10^6$  N/mm<sup>3</sup> can be used in the delamination modelling to avoid undesirable interpenetrations and numerical instabilities. In the debonding case, there is an absence of scientific papers about this. However, the penalty stiffnesses can assume values between  $10^4$  and  $10^6$  N/mm<sup>3</sup>. This parameter influences the slope of the initial linear elastic stage and the maximum obtained, but in the softening damage process the results converge (Gonçalves et al., 2000).

Regarding the approximation of the strength's values: for the delamination case,  $\sigma_{u,I}$ ,  $\sigma_{u,II}$  and  $\sigma_{u,III}$  are, respectively, equal to  $Y_{tB}$ ,  $S_{12B}$  and  $S_{13B}$  of USN150B. This happens because the delamination occurs between laminas of the composite, along the resin rich layer, and this layer is

mainly composed by the composite's matrix. Therefore, the mode I strength is similar to the composite 90° tensile strength. For the debonding situation, these values are approximated to the yield strength from de Moura et al. (2006).

The critical fracture energies chosen for the delamination of the USN150B are from Ugen et al. (2014). For the debonding of Araldite® 420 A/B, there are several sources that calculate these values such as de Moura et al. (2006), Braga et al. (2015) , Braga et al. (2018) and Mirmohammad et al. (2018), but the results are not consistent. Therefore, the values from de Moura et al. (2006) were selected. However, it is still worth mentioning that these results are from 2005 and the manufacturer may have already enhanced the formula of Araldite® 420 A/B (and with it, its fracture properties).

About the power  $\varphi$ : it is recommended for adhesives with similar critical fracture energies for the different modes of fracture, a power of  $\varphi = 1.0$  and, for adhesives with very different critical fracture energies for the different modes of fracture, it is suggested a power of  $\varphi = 1.5 - 2.0$  (Moreira et al., 2020). Consequently, in the present case, since there are similar critical fracture energies for each mode of fracture,  $\varphi = 1.0$ .

Table 4.7 resumes all the collected information about the damage parameters.

**Table 4.7** – Summary of the damage parameters chosen for USN150B and Araldite® 420 A/B.

| <b>Material</b>   | <b><math>K</math></b><br>[N. mm <sup>-3</sup> ] | <b><math>\sigma_{u,I}</math></b><br>[MPa] | <b><math>\sigma_{u,II} = \sigma_{u,III}</math></b><br>[MPa] | <b><math>G_{IC}</math></b><br>[N. mm <sup>-1</sup> ] | <b><math>G_{IIC} = G_{IIIC}</math></b><br>[N. mm <sup>-1</sup> ] | <b><math>\varphi</math></b> |
|-------------------|-------------------------------------------------|-------------------------------------------|-------------------------------------------------------------|------------------------------------------------------|------------------------------------------------------------------|-----------------------------|
| USN150B           | 10 <sup>6</sup>                                 | 61.0                                      | 70.0                                                        | 0.24                                                 | 0.74                                                             | 1.0                         |
| Araldite® 420 A/B | 10 <sup>6</sup>                                 | 27.0                                      | 27.0                                                        | 0.3                                                  | 0.6                                                              | 1.0                         |

## 4.6. FINAL REMARKS

In this chapter, the wing representative structure's geometry and constitution are explored. The elastic, elastoplastic and damaged material properties are reviewed and listed to be used later on.

It is still important to mention that to obtain the elastic properties of the carbon fibre /epoxy composite, one resorts to micromechanics-based models.

## CHAPTER 5. WING REPRESENTATIVE STRUCTURE'S ANALYSIS

---

### 5.1. INTRODUCTION

In this chapter, the whole wing representative structure (Figure 1.1) undergoes a thorough analysis. Even though there is geometrical symmetry in the parallel plane to  $Oxz$  and almost in the parallel plane to  $Oyz$  plane, due to the laminated material used, there is no material symmetry. Therefore, the simulated structure cannot be simplified using symmetry conditions. Additionally, to simplify the mesh and the analysis itself, no holes or bolts are considered.

As a result, the first step is adapting the CAD model, provided by the School of Engineering, Cardiff University, by removing all the holes and then importing it to ABAQUS software. The used version is ABAQUS 6.11 in a computer with the following characteristics: 8 CPUS I7-4790-3.66 GHz; 16 GB of RAM, 930 GB hard drive. Next, any required mechanical properties are provided (just like discussed in Chapter 4).

Then, static, and modal analyses are performed and presented in sections 5.2 and 5.3, respectively. In the static analysis, the intention is to identify areas that are most likely to fail and why and, consequently, select a critical substructure to analyse in the next chapters. On the other hand, in the modal analysis, the purpose is to know the structure's modes of vibration.

In both sections, the type of analysis, boundary conditions, FE model, solvers and requested outputs are explained. Also, initially, preliminary convergence tests are executed with the goal of evaluating the model's performance with different average element sizes and determining which are the best values for it. After, the results for the approved meshes are analysed.

Lastly, some final remarks about this whole chapter are established in section 5.4.

### 5.2. STATIC ANALYSIS

Like previously stated, the main objective of this section is to conclude about the most likely to fail regions of the wing representative structure and with that information select a new substructure to undergo the analyses of Chapter 6 and Chapter 7.

In section 5.2.1, a precursory convergence test considering only the elastic behaviour of all components and geometrical nonlinearities is elaborated to select the best mesh for the present problem. Next, in section 5.2.2, the same structure is simulated, but multiple damage mechanisms are added to the model (plasticity, Hashin's damage and Araldite's cohesive damage) to determine which is the most dangerous.

Note that, ideally, the convergence test should be developed under the same conditions, but for the sake of simplicity, all damage mechanisms are disregarded in this initial stage.

### 5.2.1. CONVERGENCE STUDY OF THE STATIC ANALYSIS ON THE WING REPRESENTATIVE STRUCTURE

In the convergence study, successive simulations with different average sizes of the FE are executed. Later on, the solution's accuracy and computational cost are compared and a fit value for the average size of the FE is determined.

The developed analysis is a static one, which means that any of the inertial terms considered in the model formulation in section 3.5.2 can be neglected. Also, it is worth mentioning that this test is developed considering only the elastic behaviour of all elements with large displacements, so neither plasticity nor damage nor contact are considered in this simulation.

The structure is considered to be sustained on one pinned and one roller support along its width, just like shown in Figure 5.1. Furthermore, two points along the  $x$  direction are fixed to avoid a rigid body motion in the  $y$  direction. There are, as well, imposed displacements of 100 mm at both top extremities of the structure – Figure 5.2.

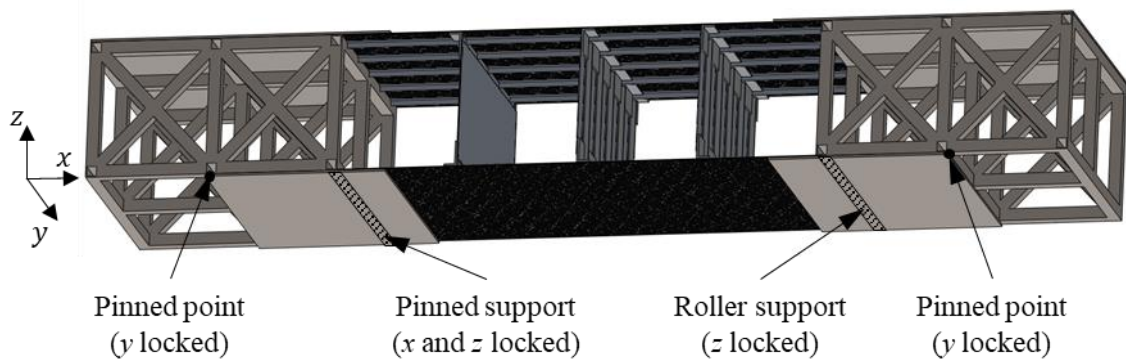


**Figure 5.1** – Boundary conditions in the static convergence study of the wing representative structure.

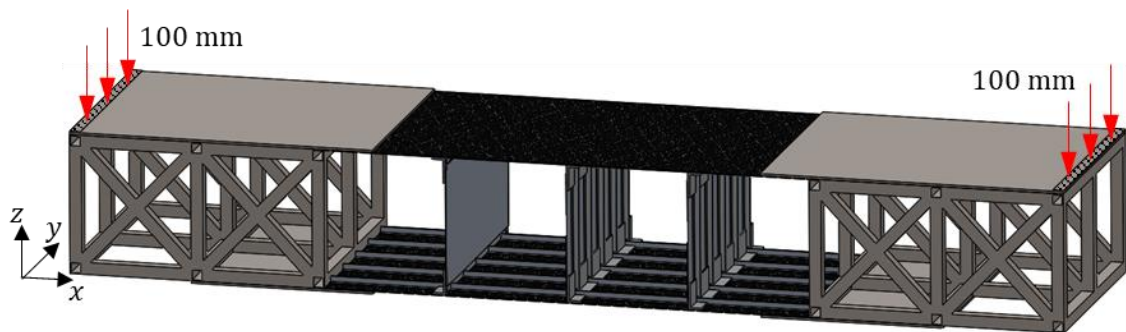


**Figure 5.2** – Imposed displacements in the static convergence study of the wing representative structure.

These boundary conditions (for the static simulation of the whole structure) are imposed by the COST project since the colleagues of the project performed an experimental analysis. Similarly, the addition of the test rig and the steel panels is inflicted by the project because there is the need to attach a test structure to the wing representative structure, which functions like a rigid body and transmits the test efforts.

Now, to perform a FE analysis it is necessary to apply a discretisation process: the idealisation of the continuous structure as an assembly of discrete FE interconnected. Therefore, regarding the FE model used, parts such as the stiffeners, the rib panels, the rib to panel joints and the attachment plates are modelled with 8-node hexahedral continuum solid elements with incompatible modes – C3D8I. These elements have the advantage of eliminating the shear locking problem and alleviating the volumetric locking one. In addition, they have a better convergence rate than tetrahedral elements and are suitable for simple rectangle-like geometries, such as the one from these parts.

On the other hand, since the composite panels are structures in which the thickness is significantly smaller than the other dimensions, structural shell FE are used to model them. More specifically, reduced integrated 8-node continuum shell elements, named SC8R, are employed. From a modelling point of view, the SC8R element looks like the C3D8I continuum solid, but its kinematic and constitutive behaviours are like the ones of conventional shell elements (first-order shear deformation theory formulation). Despite the introduction of the shell simplification, these elements bring to the model perks such as no problems with aspect ratio and lighter computational efforts. Besides, compared with conventional shell elements, they allow the evaluation of stress changes through the thickness.

As the composite panels are, in fact, laminates, they are modelled using a tool available in ABAQUS/CAE named Composite Layup. In this, there is only the need to identify the material, thickness, and orientation of each ply. After that, ABAQUS can calculate the  $\mathbb{A}$ ,  $\mathbb{B}$  and  $\mathbb{D}$  stiffness matrices necessary in a similar way to the one presented in Appendix A.

The only missing part, the test rig has a very complex geometry, with many hollow sections, so the part is subdivided and meshed using the two different elements: C3D8I elements and 10-node tetrahedral continuum solid elements – C3D10. The C3D10 element is chosen due to the existence of a fully automatic tetrahedral mesh generator that can adapt to complex geometries and so it is able to mesh the geometry when the C3D8I elements are not.

All the mentioned elements have only displacement degrees of freedom (DOF). Therefore, they can be connected directly to each other without any kinematic transition. In addition, the connection between meshed parts is established by a tie, which connects two separate surfaces together so that there is no relative motion between them. It is still important to mention that the meshes between parts can be and are in some cases non-conformal.

A schematic of these elements is presented in Figure 5.3 and any more details concerning any of them can be found in Smith (2009e).

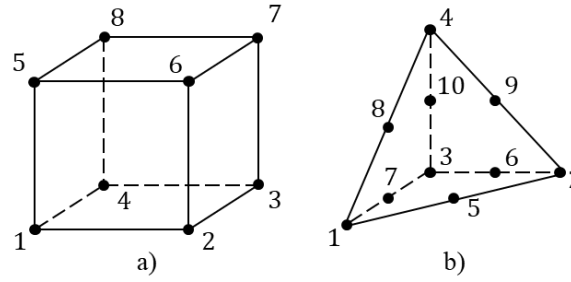


**Figure 5.3** – Schematic of the FE used: a) C3D8I and SC8R; b) C3D10.

Regarding the simulation's solver, as the analysed problem considers geometrical nonlinearities, the Newton-Raphson method (Newton's method) is used.

Since this analysis refers to a static problem, time discretisation is not required. However, to facilitate the results' convergence and evaluate the structure's response, the increasing displacement is imposed incrementally. Increments of 1 % are suggested as a starting point for the analysis. Later, the software is able to adjust it (only to smaller values) as it seems more suitable.

The requested outputs are displacements (U), stresses (S) and logarithmic strains (LE) of the whole structure, displacements in direction z in the loading areas and reaction forces (RF), also in direction z, on the supports. The values for the whole structure are recorded for every five increments and the values of the displacements and reaction forces in z are recorded for all increments. It is important to highlight that the “new” symbols for the displacements, stresses and strains are the ones assigned by ABAQUS. In the Nomenclature, there is a list of all the variables associated with ABAQUS' outputs. Also, ahead of some of these symbols the numbers 1, 2 and 3 appear to distinguish their direction.

Figure 5.4 displays the load-displacement graphics of the previously described simulation obtained using different average element sizes. The load values are the sum of all the reaction forces recorded and the displacement is from a node of the loading area.

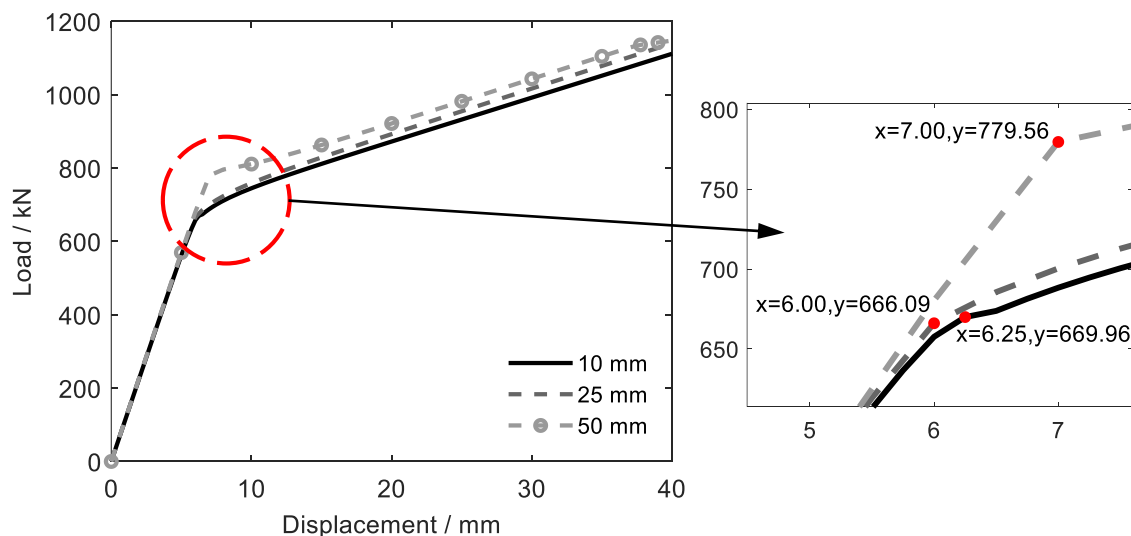


**Figure 5.4** – Load-displacement graphic, for different average element sizes, of the static convergence study on the wing representative structure.

The sudden change of the structure's stiffness is due to buckling. In the detail of Figure 5.4, the buckling point of each curve is highlighted. After the mesh is chosen, this problem is illustrated in Figure 5.6.

Additionally, Table 5.1 presents information of these simulations such as the total number of elements ( $n_e$ ) and DOF ( $n_{dof}$ ) used and spent time per increment and Table 5.2 shows the initial structure's stiffness ( $E_{in}$ ) and the buckling load. It is important to note that, in Table 5.1 the average element size is applied to every component, except the test rig where the average element size is fixed in 10 mm, because its hollow parts enforce the automatic mesh to be very refined. The measured time is presented in the time per increment format since the multiple simulations ran until different fractions of the imposed displacement. In Table 5.2, the initial stiffness is the slope of the first line of Figure 5.4 and it is obtained by a linear regression. The buckling load is read directly from Figure 5.4. The percentage values presented in Table 5.2 in parentheses are the relative difference between the calculated values for each average element size and the corresponding ones computed with an average element size of 10 mm. For any variable  $a$ , this difference can be written such as

$$\frac{a - a_{10 \text{ mm}}}{a_{10 \text{ mm}}} \times 100. \quad (5.1)$$

**Table 5.1** – Number of elements and DOF used, and time spent per increment for different average element sizes in the static analysis of the wing representative structure.

| Avg. Element Size [mm] | $n_e$   | $n_{dof}$ | Time per Increment [s] |
|------------------------|---------|-----------|------------------------|
| 10                     | 279,508 | 3,790,276 | 25.61                  |
| 25                     | 177,394 | 2,267,236 | 13.01                  |
| 50                     | 162,558 | 2,040,864 | 8.22                   |

**Table 5.2** – Initial stiffness and buckling point coordinates for different average element sizes in the static analysis of the wing representative structure.

| Avg. Element Size [mm] | $E_{in}$ [N. mm <sup>-1</sup> ] | Buckling Load [kN]   |
|------------------------|---------------------------------|----------------------|
| 10                     | 113,081.2                       | 669.96               |
| 25                     | 113,613.0<br>(+0.470%)          | 666.09<br>(−0.578%)  |
| 50                     | 114,229.6<br>(+1.016%)          | 779.56<br>(+16.359%) |

Observing the previous results, one concludes that finer meshes, that is, smaller average element sizes, imply:

- higher number of elements and DOF used and, consequently, a bigger time spent per increment;
- lower initial stiffness of the structure. However, the difference is almost negligible;

- more conservative estimates of the buckling point – they set as critical values lower loads.

Examining the differences between the results and pondering them with the time spent per increment, the maximum average element size used in any of the analyses presented throughout this chapter is 25 mm. The difference between the results is at most 0.578 %, which can be assumed as a reasonable limiting value, for such a complex analysis.

To better help in the visualisation of the FE model implemented, Figure 5.5 presents the final FE model, alongside with two labelled details of the structure.

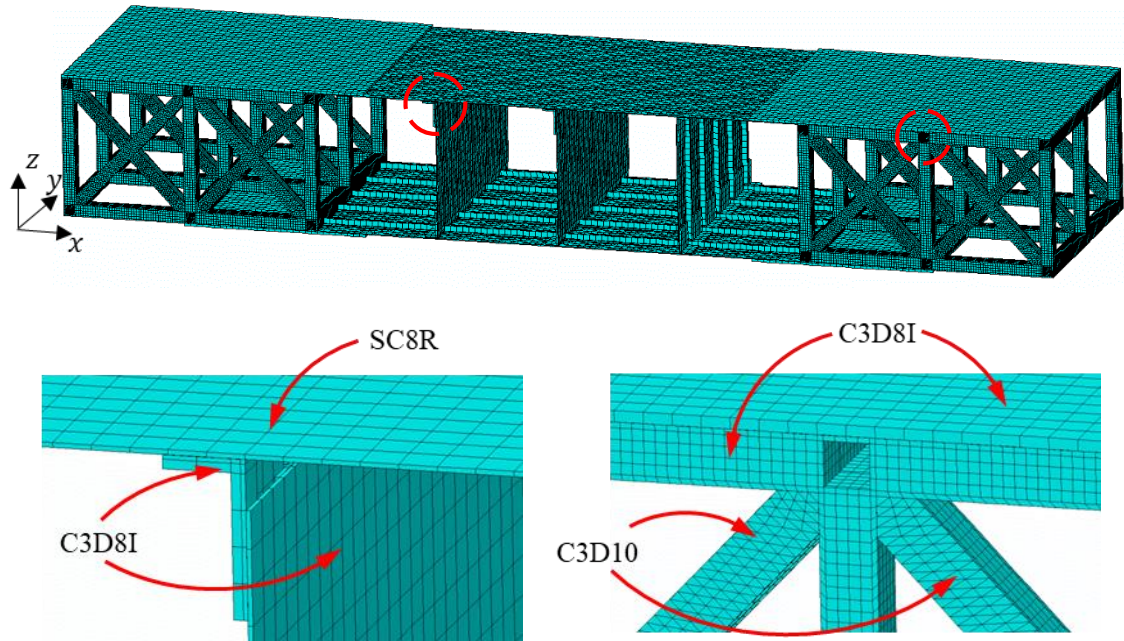


**Figure 5.5** – FE model of the wing representative structure and used elements in the static convergence study.

For the chosen mesh, Figure 5.6 (a) and Figure 5.6 (b) present the deformed structure for imposed displacements of 5 mm and 7 mm, respectively. In these, it is noticeable that the long stiffeners and the composite panels undergo buckling and jeopardise the structure's global stiffness. In the central region of the structure, the maximum displacement goes from 13.01 mm to 31.77 mm.

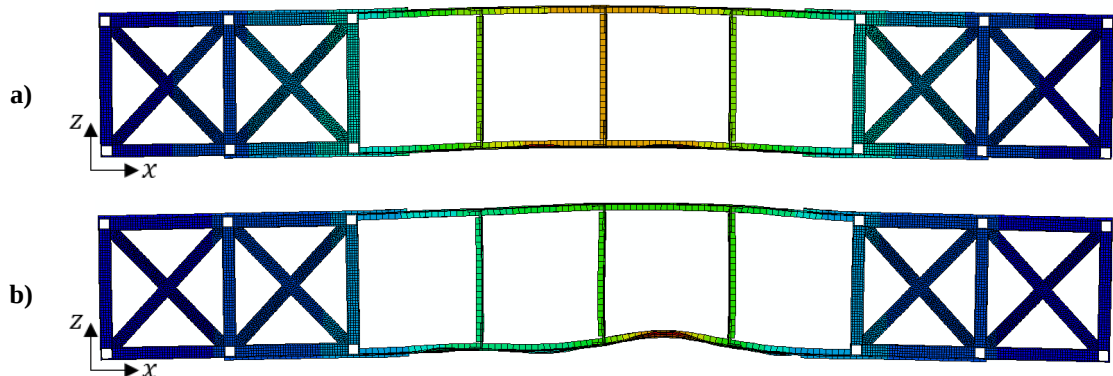


**Figure 5.6** – Displacement results of the static convergence study on the wing representative structure, for imposed displacements of: a) 5 mm; b) 7 mm. Magnified 3x.

### 5.2.2. NUMERICAL RESULTS OF THE STATIC ANALYSIS ON THE WING REPRESENTATIVE STRUCTURE

Now that the mesh is perfectly defined, a new static analysis under the same boundary conditions and loadings as the previously described in Figure 5.1 and Figure 5.2 is developed. In this, the main purpose is to discover the structure's critical regions.

To do so, besides accounting for the elastic behaviour of the materials, the elastoplastic one of aluminium 6063AT6 is considered introducing the plastic properties of this material in the simulation.

Furthermore, the Hashin's criterion is employed in the composite panels to predict anisotropic damage in fibre-reinforced composite materials and prevent four failure modes: tensile or compressive fibre failure and tensile or compressive matrix failure (Smith, 2009c). These criteria are detailed next: the tensile fibre failure occurs when  $\sigma_{11} \geq 0$  and is evaluated by

$$\left(\frac{\sigma_{11}}{X_t}\right)^2 + \frac{\tau_{12}^2 + \tau_{13}^2}{S_{12}^2} = 1, \quad (5.2)$$

and the compressive fibre failure happens when  $\sigma_{11} < 0$  and is assessed by

$$\left(\frac{\sigma_{11}}{X_c}\right)^2 = 1. \quad (5.3)$$

On the other hand, the tensile matrix failure occurs if  $\sigma_{22} + \sigma_{33} \geq 0$  and its equation is

$$\left(\frac{\sigma_{22} + \sigma_{33}}{Y_t}\right)^2 + \frac{\tau_{23}^2 - \sigma_{22}\sigma_{33}}{S_{23}^2} + \frac{\tau_{12}^2 + \tau_{13}^2}{S_{12}^2} = 1, \quad (5.4)$$

and the compressive matrix failure may occur when  $\sigma_{22} + \sigma_{33} < 0$  and is measured by

$$\left[\left(\frac{Y_c}{2S_{23}}\right)^2 - 1\right] \frac{\sigma_{22} + \sigma_{33}}{Y_c} + \left(\frac{\sigma_{22} + \sigma_{33}}{2S_{23}}\right)^2 + \frac{\tau_{23}^2 - \sigma_{22}\sigma_{33}}{S_{23}^2} + \frac{\tau_{12}^2 + \tau_{13}^2}{S_{12}^2} = 1. \quad (5.5)$$

Finally, to account for the behaviour of Araldite® 420 A/B in the connection of the stiffeners, a 0.05 mm layer of cohesive elements is added to these connections. The cohesive elements used are 8-node three-dimensional cohesive elements, called COH3D8. Just like presented in the model formulation, these elements use a traction-separation law and function in a similar way to interface elements (hence the thin thickness chosen): they assume a linear elastic traction-separation law prior to damage and failure is characterised by progressive degradation of the material stiffness, which is driven by a damage process. Thus, COH3D8 elements are ideal to model the delamination at interfaces in composites or the cohesive failure in bonded interfaces (adhesives) where the interface thickness is negligibly small, which is the case here.

In this analysis, to simplify it, since the model is already very complex, these elements only simulate the initial elastic stage (no initiation or propagation criterion is implemented). The schematisation of COH3D8 elements is the same as the one from C3D8I elements presented in Figure 5.3 and these elements also have only displacement DOF. Once more, more details concerning any of these FE can be found in Smith (2009e).

Figure 5.7 schematises the new FE model implemented and has a detail showing the elements used in the composite panel-stiffeners connection.

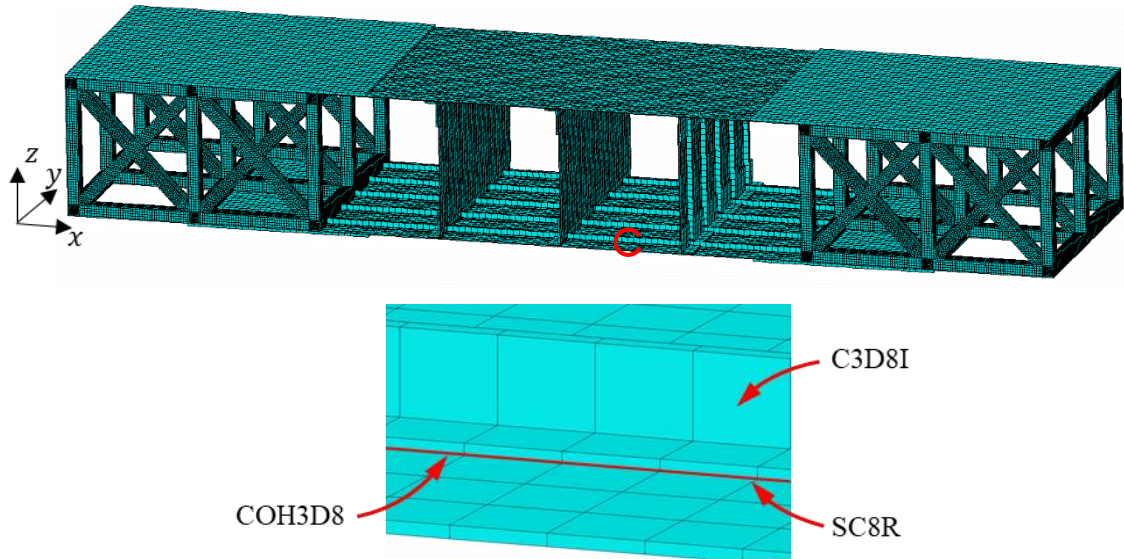


**Figure 5.7** – FE model of the wing representative structure and used elements in the static analysis.

The same solver and time discretisation methods were applied, but increments of 0.5 % are used.

For this analysis, there are additional output requests: the plastic strains (PE) in the stiffeners and the Hashin damage parameters (HSNFTCART is the fibre's tensile failure, HSNFCCART is the fibre's compressive failure, HSNMTCART is the matrix's tensile failure and HSNMCCART is the matrix's compressive failure), that indicate if the criterion is satisfied (damage parameter equal do 1) or not (damage parameter smaller than 1), in the composite panels. The results are saved for every increment.

The load-displacement graphic of this new simulation is in Figure 5.8.

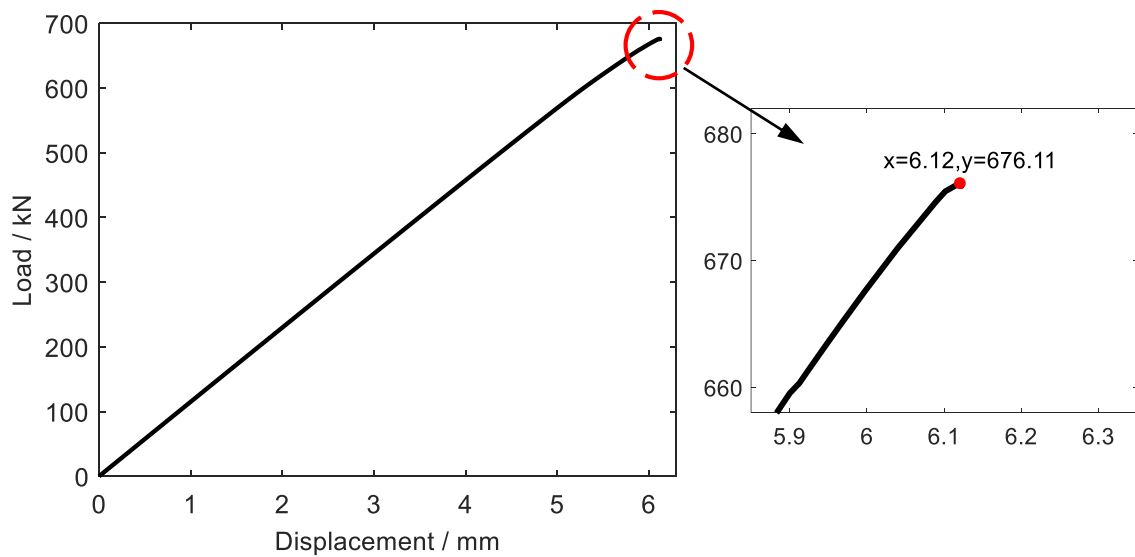


**Figure 5.8** – Load-displacement graphic of the static analysis on the wing representative structure.

The system's response is almost linear until it reaches its peak of 676.11 kN for an imposed displacement of 6.12 mm. This buckling load is very similar to the one projected in the convergence study for this mesh. After that, there is a sudden drop of the load value. In Figure 5.8, one can only observe the beginning of this drop due to convergence issues of the analysis. It is thought that this happens because of the use of plastic behaviour, combined with the beginning of buckling phenomena and the discontinuity in the load value.

The von Mises stress results for the wing representative structure and the testing one are presented in Figure 5.9 a) and Figure 5.9 b). These results correspond to the peak of the curve of Figure 5.8 and are for an imposed displacement of 6.12 mm.

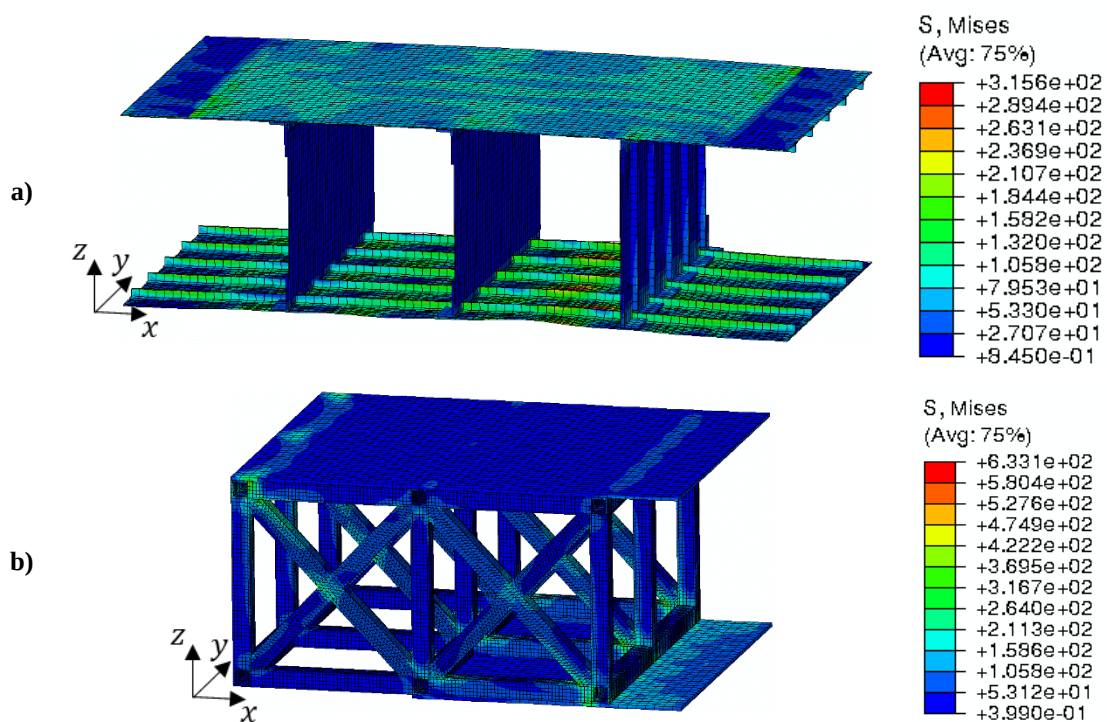


**Figure 5.9** – Stress results of the static analysis, for an imposed displacement of 6.12 mm, on the: a) wing representative structure; b) testing structure.

In the wing representative structure, the maximum von Mises stress value is 315.59 MPa on the composite panel. The vertical panels and stiffeners have very low stresses (maximum of 63.05 MPa).

On the other hand, in the testing structure a maximum of 633.09 MPa is verified. However, looking closely at Figure 5.9 b), most of the structure is coloured in blue. Therefore, one concludes that most of the structure has low von Mises stress values. The maximum results occur near sharp corners or hollow parts. Despite existing stress concentrations in these geometries, which can explain the higher stresses obtained, it is also important to state that these higher values may be exaggerated by the software's method of extrapolation, mesh size and element type. Globally, it is possible to conclude that the testing structure behaves almost like a rigid body as wanted, transmitting the test loads to the wing representative structure.

Observing now, for the same increment, the plastic strains – Figure 5.10– one can conclude that the plastic regime is reached. However, only a maximum value of 0.0507 % of plastic strain is obtained, which means that, although undesirable, this phenomenon does not significantly influence the results. This phenomenon starts for an imposed displacement of 5.7 mm.

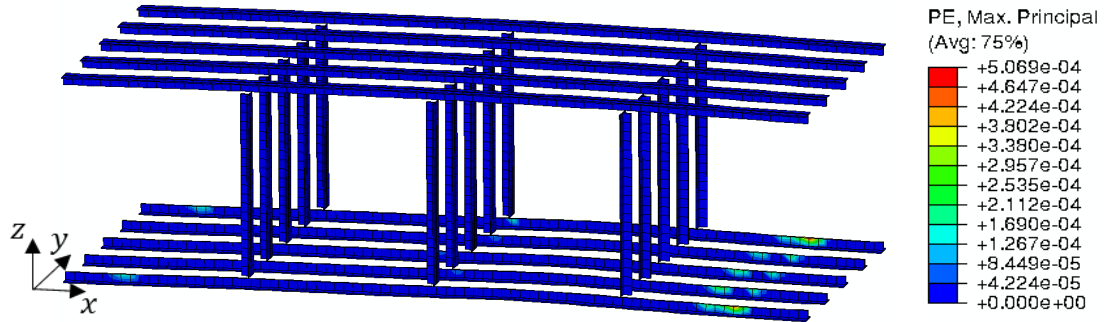


**Figure 5.10** – Plastic strain results of the static analysis, for an impose displacement of 6.12 mm, on the aluminium stiffeners.

Regarding the Hashin damage, the utmost Hashin’s damage parameters are the ones of Table 5.3. From left to right, they represent the tensile fibre failure, the compressive fibre failure, the tensile matrix failure, and the compressive matrix failure. Looking at these results, it is easy to conclude that none of these criteria is satisfied, as all parameters are smaller than 1, and that the composite panel’s damage is not the reason for the structure’s failure.

**Table 5.3** – Maximum Hashin’s damage parameters in the static analysis of the wing representative structure.

| HSNFTCRT | HSNFCCRT | HSNMTCRT | HSNMC CRT |
|----------|----------|----------|-----------|
| 0.0274   | 0.0098   | 0.1351   | 0.1925    |

Finally, the behaviour of the cohesive elements placed on the connection of the long stiffeners with the composite panel, that mimic the behaviour of the adhesive, is analysed.

Observing the highlighted region of Figure 5.11, that illustrates the cohesive elements placement, for an imposed displacement of 6.12 mm, in Figure 5.12, it is possible to acknowledge that the original thickness of these elements, 0.05 mm, has increased drastically by over 16 mm. This indicates that the adhesive suffers severe damage and even rupture, somewhere during the analysis.



**Figure 5.11** – Cohesive elements location in the stiffeners-panels interface mimicking the adhesive’s behaviour.

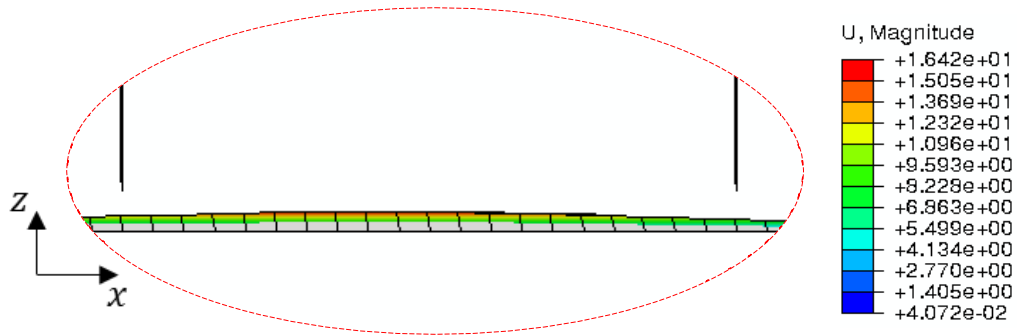


**Figure 5.12** – Detail of the displacement results of the static analysis, for an imposed displacement of 6.12 mm, on the cohesive elements.

Despite not being able to quantify this phenomenon, since no initiation criterion or propagation law are implemented, and not being able to identify the exact moment when the element's stretch is critical, this observation allows one to recognise that the bonding in this location is one of the most likely to fail first.

Looking now at the work of other WG1 colleagues, each contributor is free to approach the problem as it sees fit and, therefore, every result is different and independent. Comparing the load-displacement graphic of other colleagues with the one from Figure 5.8, it is important to emphasise that the initial slopes in all the approaches are very similar. However, the critical loads vary significantly, mainly due to the adoption of different failure criteria.

The experimental static test was carried out in Cardiff School of Engineering by Dr. Stephen Grigg, Faisal Almudaihesh and Prof. Rhys Pullin. The structure fails in the composite panel-stiffener connections: firstly, two stiffeners debond at around 250 kN, next, other two debond at 275 kN and 300 kN, respectively. Therefore, as identified in the run simulation, the crucial regions of the wing representative structure are the composite panel-stiffener connections.

Using this information, one can reduce the object of future analyses to a simpler substructure with resembling problems. This substructure is composed by a quarter of the composite panel reinforced with five “T” aluminium stiffeners, such as displayed in Figure 5.13. This substructure is the one that undergoes thorough analyses in the next chapters.

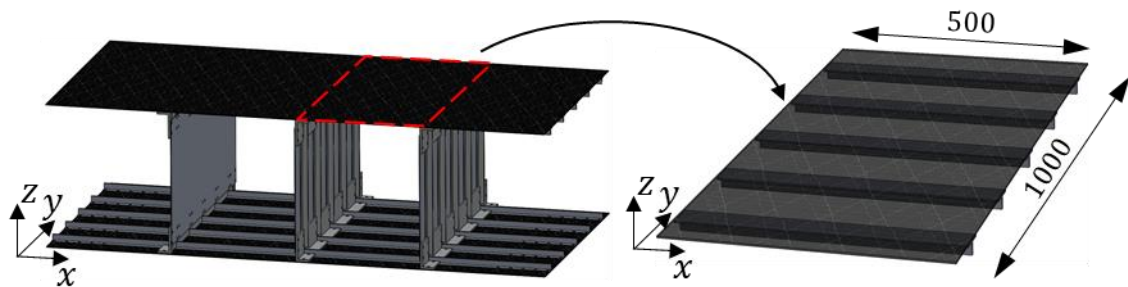


**Figure 5.13** – Original wing representative structure to the left and the new chosen substructure on the right.

### 5.3. MODAL ANALYSIS

The purpose of this section is to perform a modal analysis of the wing representative structure of Figure 1.1.

This analysis is also based on an initial convergence test to select the mesh element's average size – section 5.3.1. After having this defined, in section 5.3.2, the structure's modes of vibration are observed.

#### 5.3.1. CONVERGENCE STUDY OF THE MODAL ANALYSIS ON THE WING REPRESENTATIVE STRUCTURE

Once more, the first step of the analysis is to do a convergence test that determines the best average element size.

In this test, no nonlinearities are considered. The reason is that the software is incapable of considering them when solving an eigenvalue problem. That means that no geometric nor material nonlinearities are accounted for in this analysis.

The wing representative structure is submitted to free boundary conditions, since the real ones are not known and the eigenvalue extraction method used is Lanczos due to its general capabilities, even though it has a high computational cost (Smith, 2009a). Finally, the requested outputs are the first 11 natural frequencies of the system.

The implemented FE model is developed accordingly with section 5.2.1. However, since the analysis is not static, the test rig and attachment plates are not simulated.

Table 5.4 presents the collected information about these simulations: total number of elements and DOF used and spent time; and Table 5.5 shows the first 3 natural frequencies obtained (excluding the ones from rigid body motion) for each average element size and its respective relative difference comparing with the calculated values for a 5 mm average element size.

**Table 5.4** – Number of elements and DOF used, and time spent for different average element sizes in the modal analysis of the wing representative structure.

| Avg. Element Size [mm] | $n_e$   | $n_{dof}$ | Time [s] |
|------------------------|---------|-----------|----------|
| 5                      | 326,420 | 4,178,998 | 2228.6   |
| 10                     | 83,502  | 1,096,544 | 440.45   |
| 25                     | 14,988  | 214,784   | 82.51    |
| 50                     | 4,952   | 80,572    | 40.86    |

**Table 5.5** – First five natural frequencies (rather than 0) obtained for different average element sizes in the modal analysis of the wing representative structure.

| Avg. Element<br>Size [mm] | $\omega_i$ [rad. s <sup>-1</sup> ] |            |            |           |           |
|---------------------------|------------------------------------|------------|------------|-----------|-----------|
|                           | $i = 7$                            | $i = 8$    | $i = 9$    | $i = 10$  | $i = 11$  |
| 5                         | 890.08                             | 1306.02    | 1542.40    | 2243.22   | 2357.07   |
| 10                        | 914.64                             | 1343.91    | 1561.87    | 2247.37   | 2248.25   |
|                           | (+2.760%)                          | (+2.901%)  | (+1.263%)  | (+0.185%) | (+0.181%) |
| 25                        | 940.28                             | 1377.71    | 1582.42    | 2259.12   | 2375.23   |
|                           | (+5.640%)                          | (+5.489%)  | (+2.595%)  | (+0.709%) | (+0.770%) |
| 50                        | 1101.63                            | 1566.08    | 1707.08    | 2293.68   | 2399.67   |
|                           | (+23.768%)                         | (+19.912%) | (+10.677%) | (+2.249%) | (+1.807%) |

From these results, it is possible to conclude that, once again, the use of finer meshes implies a higher number of elements and DOF used and, consequently, a bigger time spent. In addition, using a finer mesh, the natural frequencies have an asymptotic convergence and the biggest relative differences occur for the first 2 natural frequencies.

Weighing the accuracy ensured by each average element size and its computational cost, the maximum average element size chosen for any of the modal analyses of the wing representative structure is 10 mm.

### 5.3.2. NUMERICAL RESULTS OF THE MODAL ANALYSIS ON THE WING REPRESENTATIVE STRUCTURE

Next, the results of the 10 mm average element size analysis are presented. The first 6 modes of vibration are rigid body modes since the boundary conditions are free. Figure 5.14 to Figure 5.18 present the next 5 modes of vibration of the system. The mode shapes undergo a mass normalisation.

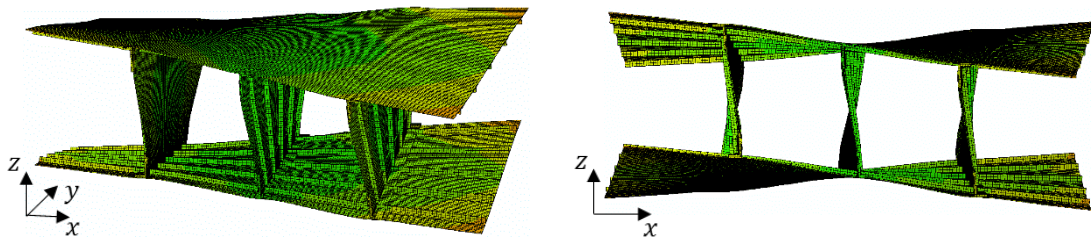


**Figure 5.14** – 7<sup>th</sup> mode shape of the wing representative structure for  $\omega = 145.57$  rad. s<sup>-1</sup>.

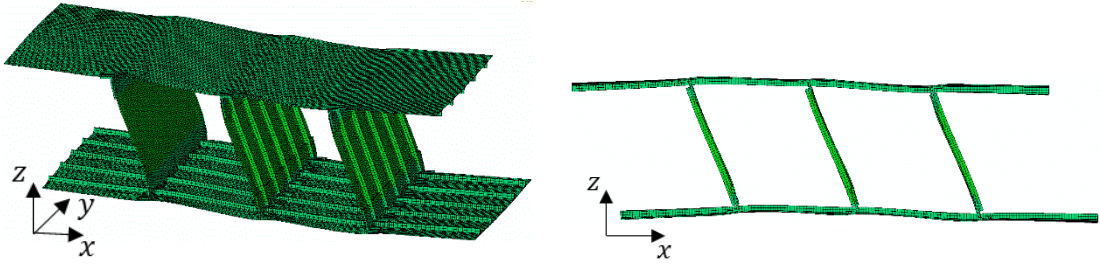


**Figure 5.15** – 8<sup>th</sup> mode shape of the wing representative structure for  $\omega = 213.89 \text{ rad. s}^{-1}$ .

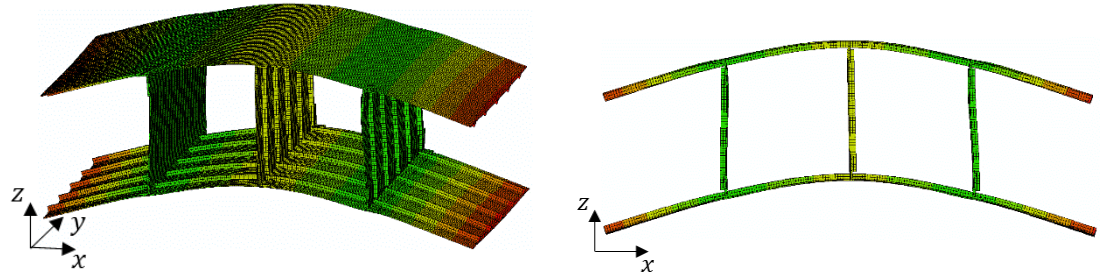


**Figure 5.16** – 9<sup>th</sup> mode shape of the wing representative structure for  $\omega = 248.58 \text{ rad. s}^{-1}$ .

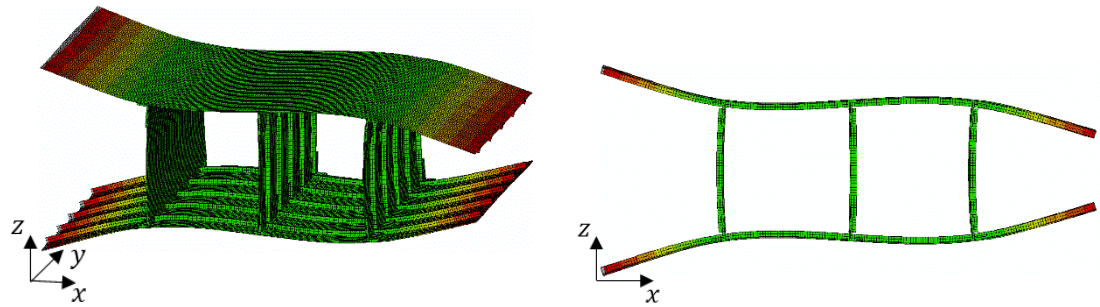


**Figure 5.17** – 10<sup>th</sup> mode shape of the wing representative structure for  $\omega = 357.68 \text{ rad. s}^{-1}$ .

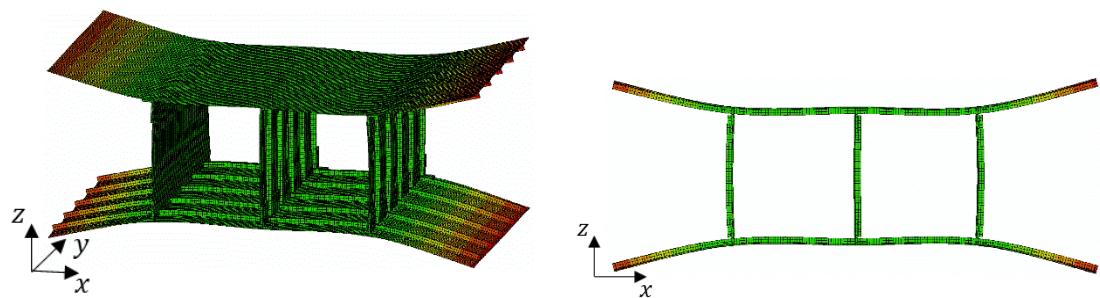


**Figure 5.18** – 11<sup>th</sup> mode shape of the wing representative structure for  $\omega = 375.82 \text{ rad. s}^{-1}$ .

Observing the modes of vibration, it is possible to deduce that the 7<sup>th</sup> mode of vibration (Figure 5.14), is a torsional one and that the remaining ones (Figure 5.15 to Figure 5.18) are flexional ones.

Additionally, observing each one is possible to say that in:

- the 7<sup>th</sup> mode shape, the composite panels' extremities vibrate in phase opposition;
- the 8<sup>th</sup> mode shape, the composite panels themselves vibrate in phase opposition. The horizontal central lines of the vertical aluminium panels are nodal lines;

- the 9<sup>th</sup> mode shape, the composite panels' extremities vibrate in phase and in opposition with the central region. There are two lines in each composite panel that are nodal lines;
- the 10<sup>th</sup> mode shape, the composite panels are divided in four "parts" in which each one is in phase opposition with the adjacent parts. There are three lines that work as nodal lines, nearly coincident with the coupling with the three aluminium panels;
- the 11<sup>th</sup> mode shape, the composite panels are divided in five "parts", being that the extremities and the central region vibrate in phase. There are four lines in each composite panel that are nodal lines.

## 5.4. FINAL REMARKS

First, in this chapter, a static analysis of the wing representative structure accounting for elastoplastic behaviour of the stiffeners, anisotropic damage of the composite panel and the cohesive behaviour of Araldite® 420 A/B in the connection of the stiffeners to the panels is developed. This analysis allows to identify the layer of Araldite® 420 A/B in the composite panel-long stiffeners connection as the critical location of the structure and to choose the substructure from Figure 5.13 for the progressive damage and dynamic analyses of the next chapters.

Next, from the modal analysis of the structure one obtains and observes its modes of vibration.



# CHAPTER 6. SUBSTRUCTURE'S PROGRESSIVE DAMAGE ANALYSIS

---

## 6.1. INTRODUCTION

Throughout this chapter, a substructure of the whole wing representative structure (Figure 5.13) is submitted to a progressive damage analysis. This substructure is selected to simplify and speed up this analysis, but it is also expected to present similar fragilities.

Once more, there are geometrical symmetries but not material ones. Thus, the substructure is not simulated using symmetry conditions. Furthermore, no holes or bolts are considered for simplicity's sake. So, the original CAD model is adapted and then imported into ABAQUS software.

The progressive damage analyses developed are explored in section 6.2. The main goal of these is to understand the location most prone for a damage occurrence and how that phenomenon evolves.

In all simulations, the type of analysis, boundary conditions, FE model, solvers and requested outputs are once again well described and explored. The results of these analyses are only scanned after the mesh has been approved in a convergence test.

## 6.2. PROGRESSIVE DAMAGE ANALYSIS

To achieve the purpose of this section, firstly, a convergence test considering only the elastic behaviour of all components and geometrical nonlinearities is developed to select the mesh to use in the next analyses – section 6.2.1. These results also help to recognise the interfaces more prone to damage and which are subject to the progressive damage analysis itself.

On the other hand, section 6.2.2 simulates the same structure, adding cohesive elements to the analysis, and concludes about the critical interface and how the damage phenomenon occurs. In section 6.2.3, to deepen the previous analysis an initial flaw is added and the differences in the structure's response are observed.

### 6.2.1. CONVERGENCE STUDY OF THE PROGRESSIVE DAMAGE ANALYSIS ON THE SUBSTRUCTURE

In this analysis, an adequate value for the average size of the FE is set by running multiple simulations with different average element sizes.

The static analysis carried out is a three-point bending one, with an imposed displacement of 50 mm at the central line of the structure. Figure 6.1 represents all the boundary conditions applied in detail. Note, that the two points fixed along the  $x$  direction are to avoid the rigid body motion in the  $y$

direction. This test is developed considering only the elastic behaviour of all components and geometrical nonlinearities.

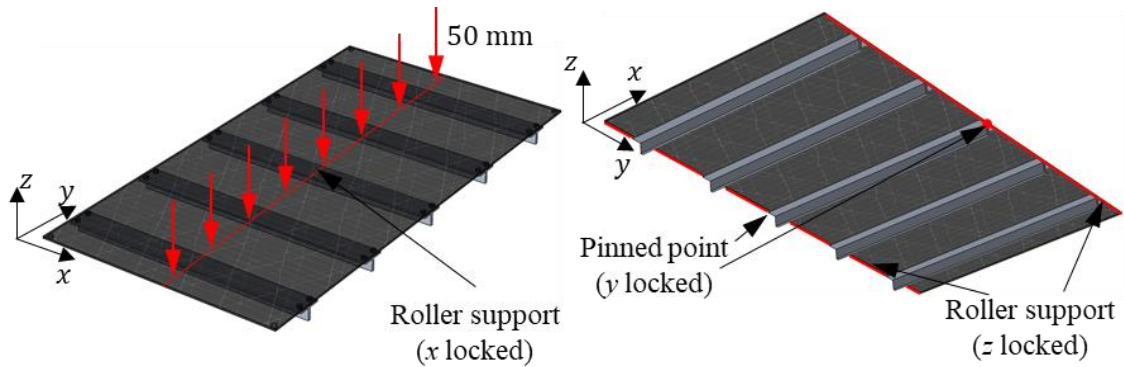


**Figure 6.1** – Boundary conditions and imposed displacements in the progressive damage convergence study of the substructure.

Just like in Chapter 5, the nonlinear problem is solved by Newton-Raphson method and the load is applied incrementally. The initial increment used is 0.1 %. The requested outputs are displacements and reaction forces, in the  $z$  direction, on the supports. The values are recorded for every increment.

The FE model used is very similar to the one presented in the last chapter. The stiffeners are discretised by C3D8I elements and the composite panel by SC8R ones. The mechanical properties of the composite panel are also implemented using the Composite Layup. The final meshes used in this chapter are conformal.

The generated meshes have two regions: a coarser one, localised near the supports, and a refined one, localised on the central region of the components with an extension of 200 mm. The refined area uses an average element size that is half the one from the coarser area. After the mesh is chosen, the FE model is illustrated in Figure 6.3.

Figure 6.2 presents the load-displacement graphics that result from the previously described simulation, for multiple meshes: 5, 10 and 20 mm average element sizes in the coarser area and 2.5, 5 and 10 mm average element sizes in the refined one. To simplify the reference to each of these meshes, they are referred to by the size of elements in the coarser region.

Table 6.1 lists the additional information of the simulations: the total number of elements and DOF used, computational time and the load applied to the system for the 50 mm displacement. The percentage values presented in parentheses are the relative difference between the calculated values for each average element size and the corresponding ones computed with an average element size of 5 mm.

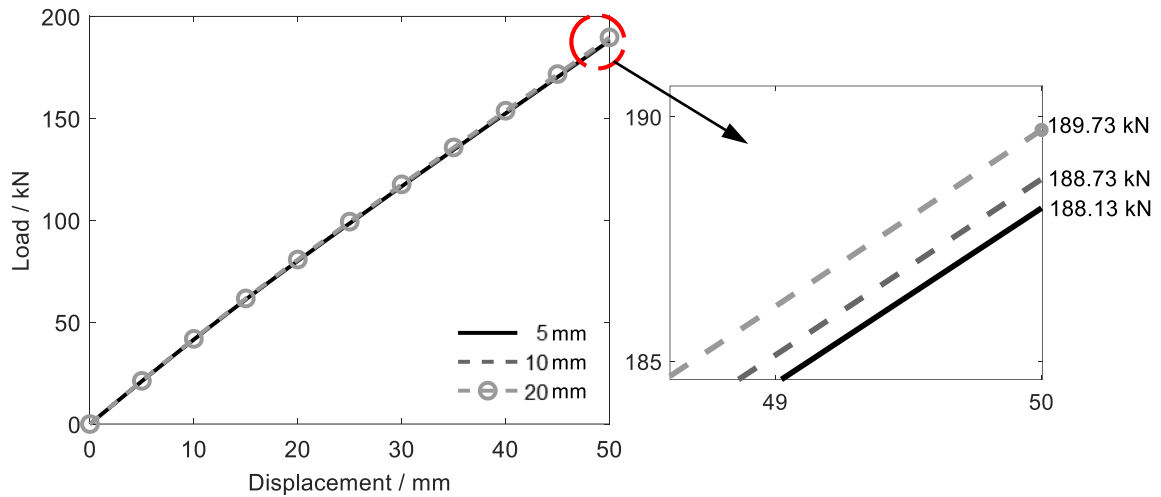


**Figure 6.2** – Load-displacement graphic, for different average element sizes, of the progressive damage convergence study on the substructure.

**Table 6.1** – Number of elements and DOF used, time spent, and maximum load for different average element sizes in the progressive damage analysis of the substructure.

| Avg. Element Size [mm] | $n_e$  | $n_{dof}$ | Time [s] | Load [kN]           |
|------------------------|--------|-----------|----------|---------------------|
| 5                      | 36,400 | 306,936   | 7384     | 188.13              |
| 10                     | 10,360 | 92,904    | 2127     | 188.73<br>(+0.315%) |
| 20                     | 2,484  | 23,670    | 545      | 189.73<br>(+0.849%) |

From these analyses, it is possible to deduce that, just like mentioned before, finer meshes are related with higher number of elements, DOF and analysis time. Regarding the load applied to the system, it decreases for meshes with smaller average element sizes.

From Figure 6.2, it is possible to conclude that the acquired results for each of these simulations are very similar and the load-displacement relation is linear. They diverge slightly, but the load necessary for a 50 mm displacement differs only 0.849% between them. Consequently, taking into account the accuracy of the results and the time spent for each simulation, the best mesh is the one with elements of 20 mm near the edges and 10 mm ones in the centre region.

Nevertheless, the chosen mesh is the one with elements of 10 mm near the edges and 5 mm ones in the centre region to assure that when the cohesive elements are applied the results continue to be accurate – Figure 6.3.

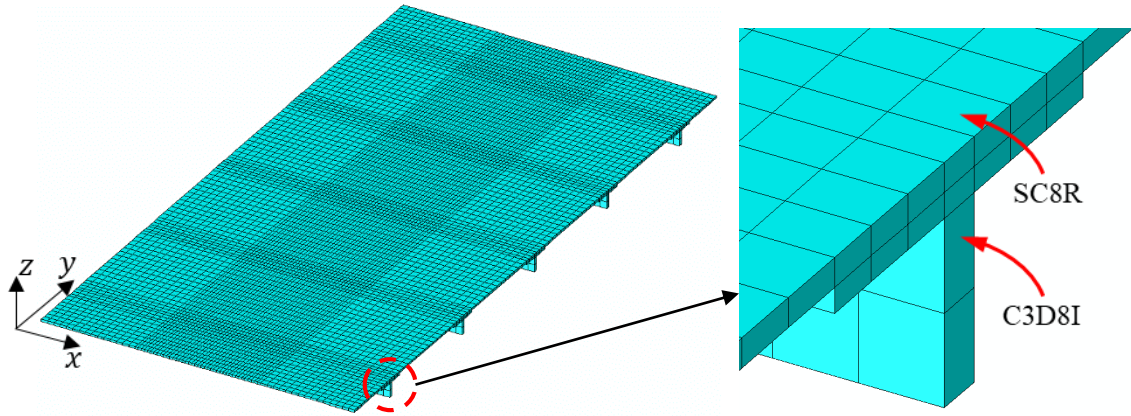


**Figure 6.3** – FE model of the substructure and used elements in the progressive damage convergence study.

Now, paying attention to the results for the chosen mesh, damage prone locations can be identified. In this work, it is considered that damage may occur between any plies of the laminated composite material or in the interface between the composite panel and the stiffeners. The critical interfaces are characterised by high stresses and/or a big mismatch bending (significant difference of the stiffnesses in the bending direction of the two bodies in contact).

To support the decision of which interfaces are critical, Figure 6.4 schematises the von Mises stress distribution on the substructure for an imposed displacement of 50 mm. In it, it is noticeable that, for all components, the von Mises stresses are higher in the centre region, away from the supports. Thus, damage is very likely to initiate in this region, which corroborates the choice of refining the mesh in this area. Also, the higher stress occurs in the stiffeners and is 3662.48 MPa. This location is likely to suffer yielding.

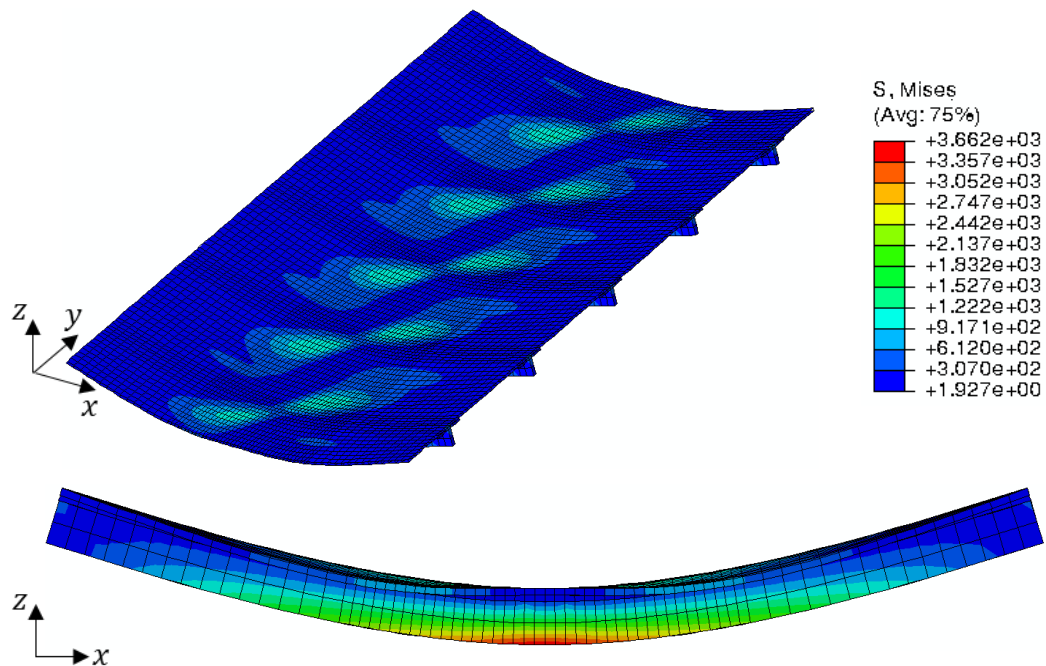


**Figure 6.4** – Von Mises stress results of the progressive damage convergence study, for an imposed displacement of 50 mm, on the substructure.

On the other hand, Figure 6.5 a) and Figure 6.5 b) present the von Mises stress distribution through the thickness of the substructure in its centre. In Figure 6.5 a), the dashed line separates the stresses in the composite panel (above the line) and in the stiffener (below the line). In Figure 6.5 b), the stress distribution through the thickness of the composite panel is detailed and the dashed lines represent the interfaces between the different subtitled plies.

Additionally, note that the von Mises equivalent stress is not usually employed in composite materials. However, even though the ABAQUS software possesses other failure criteria, such as the maximum stress, the Tsai-Hill or the Tsai-Wu criterion, it is not capable of displaying their equivalent stress distribution (Smith, 2009b).

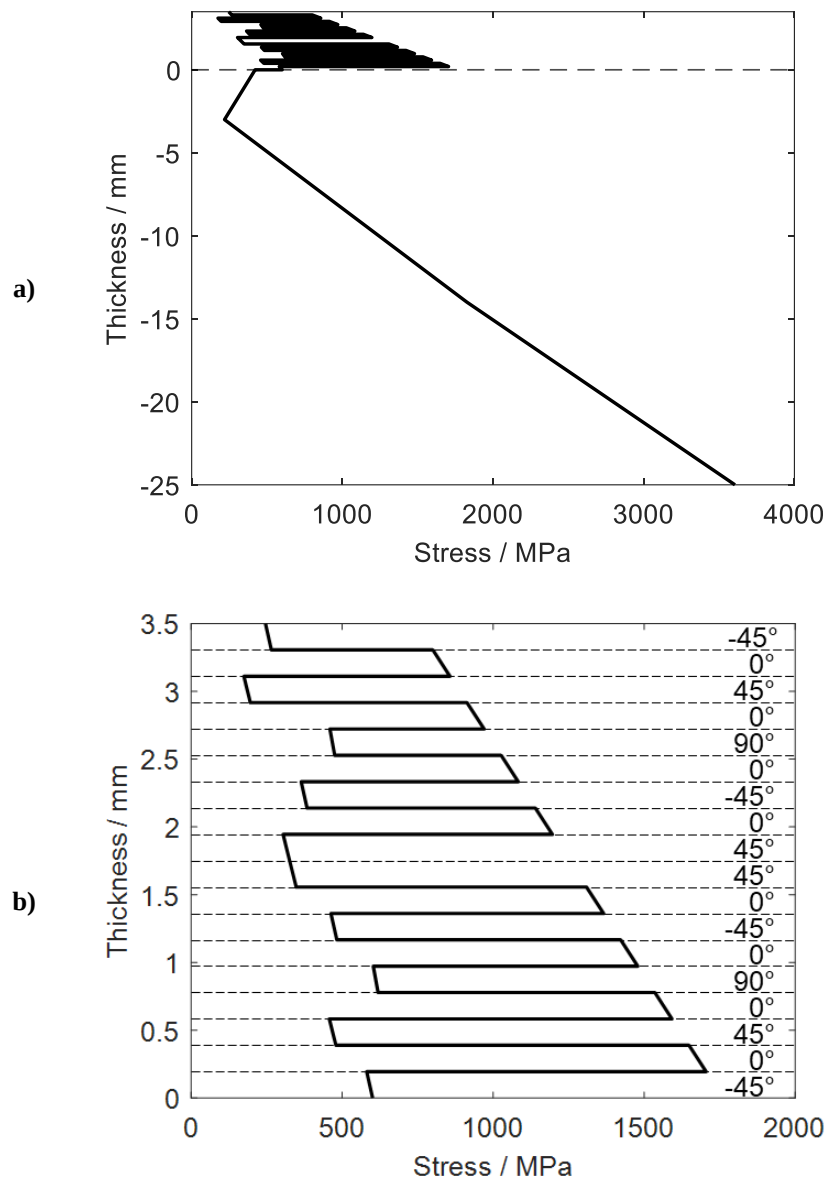


**Figure 6.5** – Von Mises stress results of the progressive damage convergence study, for an imposed displacement of 50 mm, through the thickness: a) of the substructure (composite panel + stiffener) in the centre region; b) of the composite panel.

Observing Figure 6.5 a), one concludes that the higher von Mises stress is in fact located in the stiffener and that the stresses in the composite panel are by far smaller than the ones on the stiffener. Paying attention to Figure 6.5 b), it is obvious that the plies closer to the stiffener are submitted to higher stresses and that the ones enduring the bigger efforts are in the  $0^\circ$  direction.

From this stress analysis, the critical interface submitted to the highest stresses is the  $-45^\circ/0^\circ$  one, closer to the stiffener.

Regarding the mismatch bending between the composite's plies, the  $0^\circ/90^\circ$  interfaces are the crucial ones. Since the  $0^\circ/90^\circ$  interface closer to the stiffener undergoes bigger efforts, it is selected. Lastly, the composite panel-stiffener interface also has a big mismatch bending due to the big difference of stiffnesses of the parts.

Therefore, the crucial interfaces (where the cohesive elements need to be placed) are the composite panel-stiffener one, where the cohesive elements mimic the behaviour of Araldite® 420 A/B, and the ones between the  $-45^\circ/0^\circ$  plies and the  $0^\circ/90^\circ$  plies, where the elements simulate a delamination.

### **6.2.2. NUMERICAL RESULTS OF THE PROGRESSIVE DAMAGE ANALYSIS ON THE SUBSTRUCTURE**

The goal of the following analysis is to point out the most probable way of failure of the substructure considered. To do so, aluminium 6063AT6 is considered an elastoplastic material and in the previously identified crucial interfaces are placed cohesive elements like the ones described in Chapter 5. These elements are ideal to model the delamination at interfaces in composites or the cohesive failure in adhesives, where the interface thickness is negligibly small, which are the problems here addressed. The ones used in the middle of the composite panel have a thickness of 0.02 mm and the ones between the panel and the stiffeners have 0.05 mm. Once again, as ABAQUS treats these elements as if they were interface elements, its thickness is pretty much irrelevant. The composite's fibre or matrix failures are not accounted for in this simulation, since in previous analyses there is no evidence that this is a possible failure mechanism.

Material models exhibiting softening behaviour and stiffness degradation often lead to severe convergence difficulties. Therefore, to overcome some of these convergence difficulties, a viscous regularisation process of the constitutive equations is applied. In ABAQUS software, the only extra information required is the value of the viscosity parameter  $\mu$  that represents the relaxation time of the viscous system. Using a viscous regularisation with a small value of the viscosity parameter (must be small compared to the characteristic step increment) usually helps improve the rate of convergence of the model in the softening regime, without compromising results (Smith, 2009d).

For all analyses containing cohesive elements along this chapter, the viscosity parameter used is  $\mu = 10^{-6}$  s. This value is big enough so that there are no convergence issues, but small enough to not interfere in the obtained results.

The imposed boundary conditions, as well as the solver used, are still the same. However, the imposed displacement is now 200 mm. The initial increment is 0.1 % and the requested output results

are the same as the convergence study ones, plus plastic strains in the stiffeners and damage parameters (SDEG) in the cohesive elements. These damage parameters are calculated just like shown in Equation (3.53).

The mesh used in this analysis is displayed in Figure 6.6.

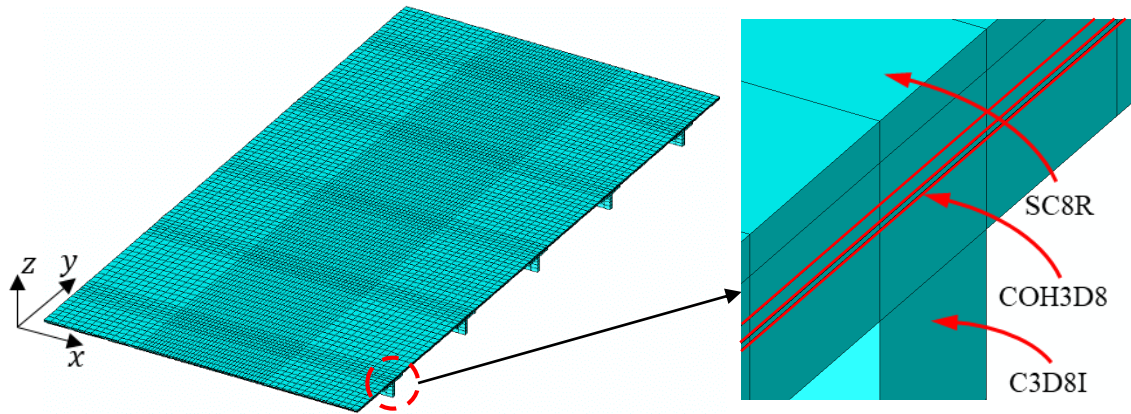


**Figure 6.6** – FE model of the substructure and used elements in the progressive damage analysis.

In Figure 6.7, the load-displacement graphic for this simulation is shown. The letters intend to separate different stages of this response.

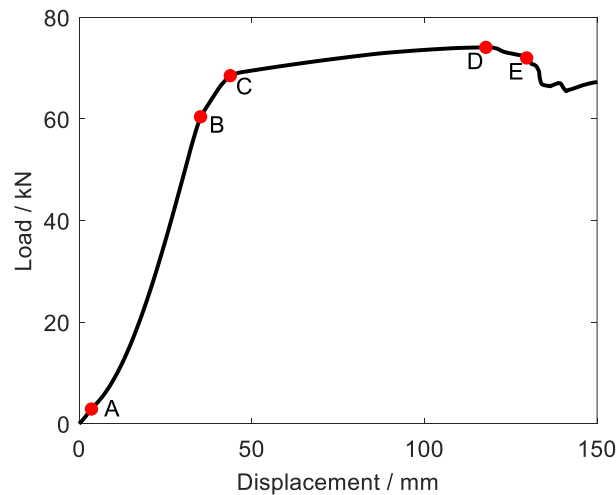


**Figure 6.7** – Load-displacement graphic of the progressive damage analysis on the substructure.

Firstly, until point A, the system's response is linear. In point A, with coordinates (3.6 mm, 2.889 kN), the stiffeners start yielding and the damage parameter becomes different than 0, which means that the material's degradation process has already begun. This damage is located only in the composite panel-stiffeners interface.

Then, there is a change of the system's stiffness, and the response is once again linear until point B, with coordinates (35.2 mm, 60.425 kN). In this, the maximum plastic strain verified is now  $PE = 6.891\%$  and the maximum damage parameter is  $SDEG = 0.919$ . This increase of stiffness is

unexpected since the appearance of plasticisation and damage should decrease it. A variable that might be increasing it is the approximation of the supports during the analysis.

These values continue to increase, causing a nonlinear response of the system, until point C, with coordinates (43.8 mm, 68.505 kN), where the maximum plastic strain verified is  $PE = 8.199\%$  and the maximum damage parameter is  $SDEG = 0.936$ . From this point on, the stiffeners should rupture (in the region that has reached the limit value – Figure 6.8) and affect the overall substructure's stiffness causing a sudden drop of the load applied. In this figure, the legend only starts at  $PE = 6.0\%$ , so that the limit region is highlighted in red.

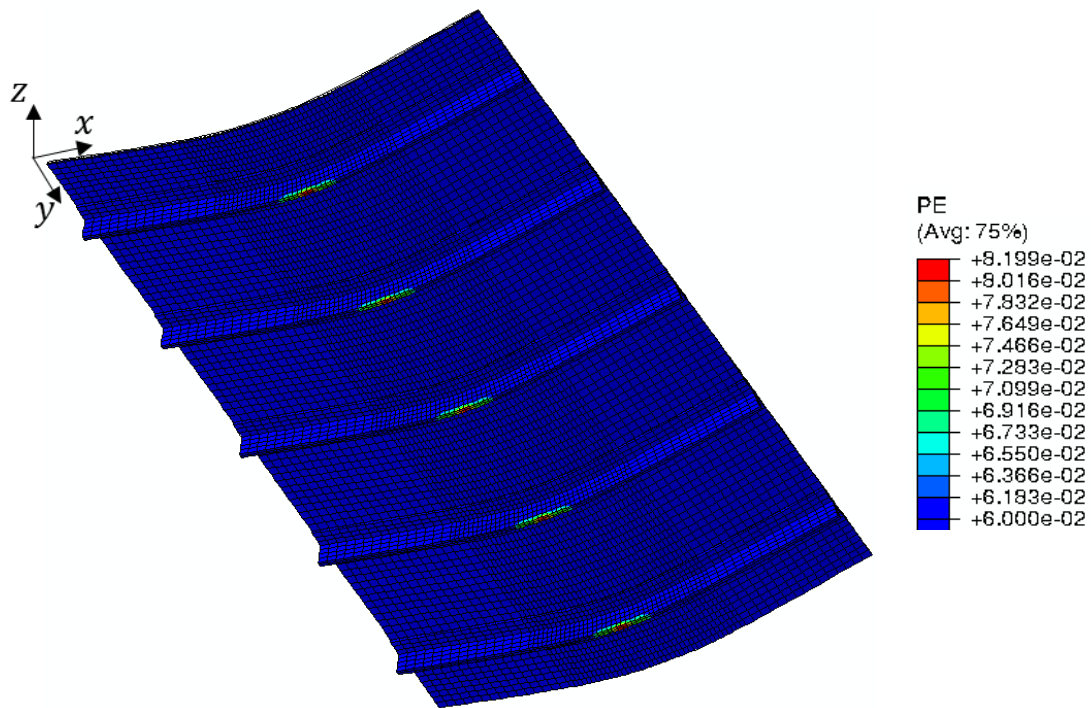


**Figure 6.8** – Plastic strain results of the progressive damage analysis, for an imposed displacement of 43.8 mm, on the substructure.

However, the load keeps increasing, because the ABAQUS software assumes that after the last point of the hardening curve the material becomes perfectly plastic. It increases until point D, with coordinates (117.8 mm, 74.085 kN), where the applied load is maximum. In it, the maximum plastic strain and damage parameter are, respectively,  $PE = 18.657\%$  and  $SDEG = 0.980$ .

Next, the load slowly decreases as the previous variable continues to increase, until point E, with coordinates (129.6 mm, 72.002 kN). At this point, the maximum damage parameter is  $SDEG = 1.0$  for the first time. Figure 6.9 a) and Figure 6.9 b) display the plastic strains in the stiffeners and the damage parameters in the composite panel-stiffeners interface (where they are maximum) for point E. The legends start at  $PE = 6.0\%$  and  $SDEG = 0.90$  to highlight the critical or almost critical regions.

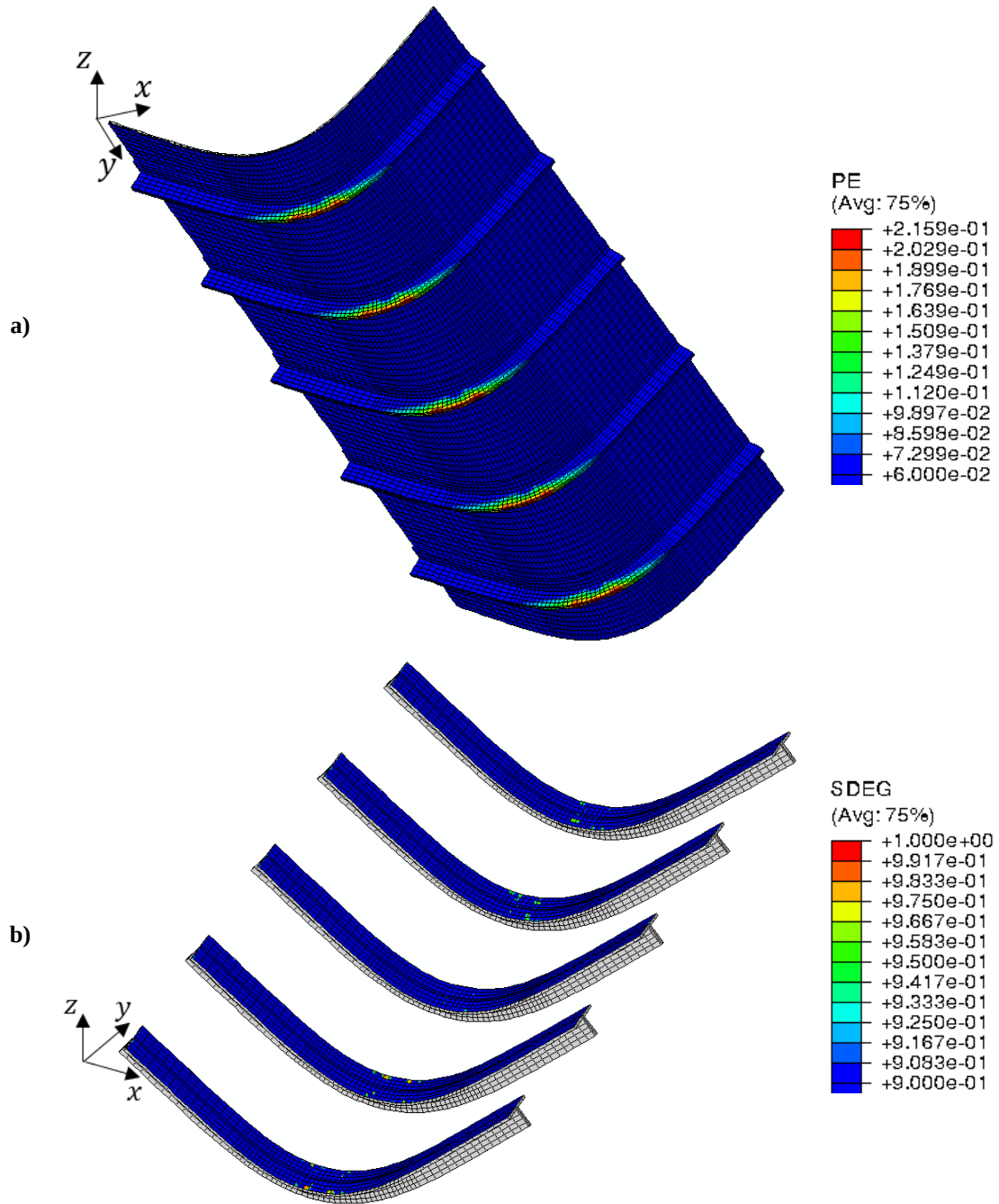


**Figure 6.9** – Plastic strain (a) and damage parameter (b) results of the progressive damage analysis, for an imposed displacement of 129.6 mm, on the substructure.

Right after, the rupture of the cohesive layer extends in a way that deeply affects the overall stiffness of the substructure and the applied load drops. Figure 6.10 a) and Figure 6.10 b) show the evolution of the outputs when the load is decreasing.

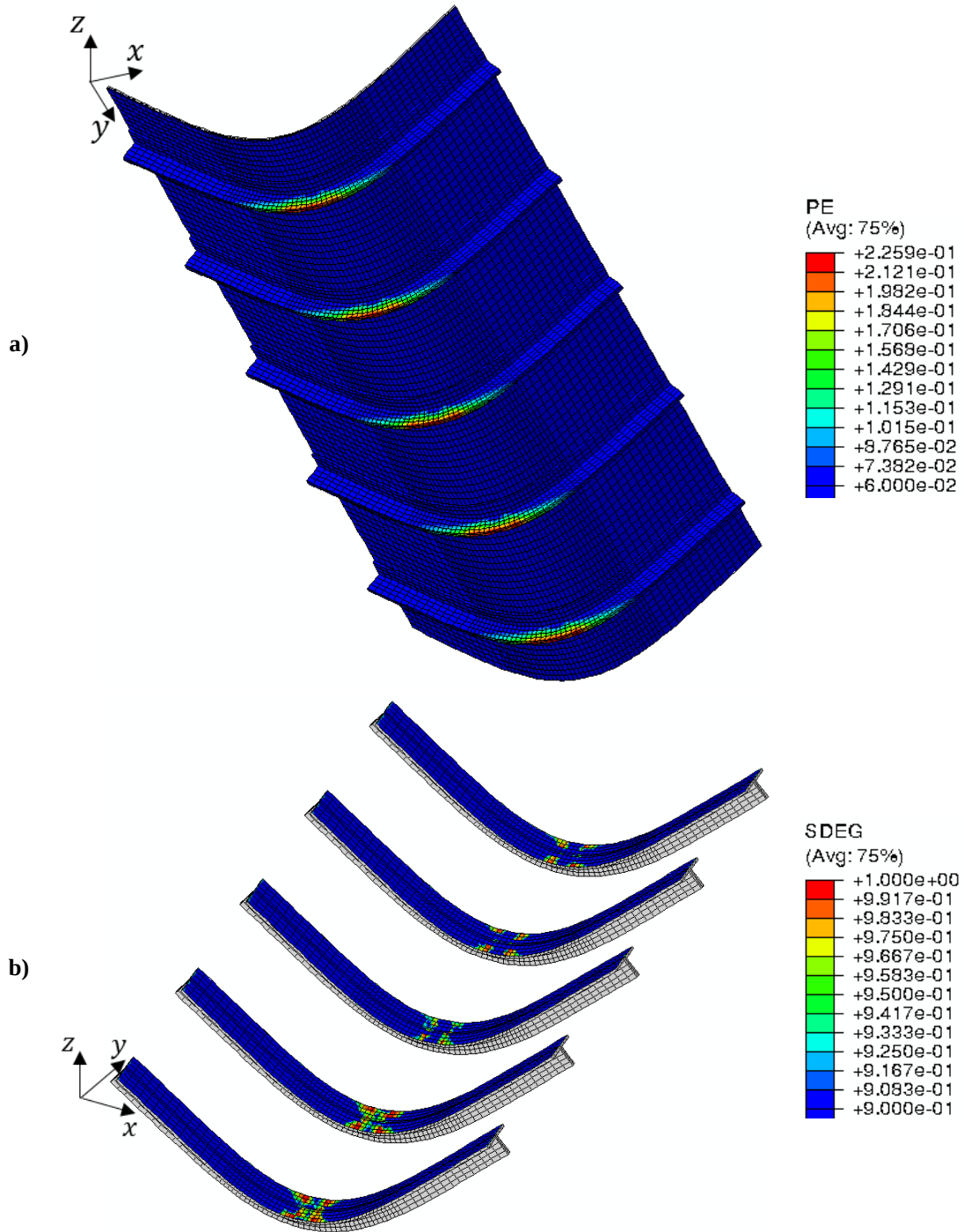


**Figure 6.10** – Plastic strain (a) and damage parameter (b) results of the progressive damage analysis, for an imposed displacement of 132.4 mm, on the substructure.

As expected, the plastic strains kept increasing during the rupture, significantly enlarging the area totally plasticised. In addition, the area with softened properties or in rupture is much larger, which causes the detachment of the components and highly contributes to the loss of stiffness of the overall substructure.

In short, in the ran simulation, the substructure fails due to yielding and cohesive damage, combined. In it, it is possible to identify the critical interface as the composite panel-stiffener one since

it is the only one where the damage parameter reaches the value 1.0. Thus, the debond of the Araldite® 420 A/B layer is more likely to occur than the composite panel's delamination.

However, it is still important to highlight that this analysis does not faithfully represent the real conditions of the problem, and, in fact, it is expected that the substructure fails mainly due to the yielding and rupture of the stiffeners. The softening process assumes a much smaller importance in this way of failure.

### 6.2.3. NUMERICAL RESULTS OF THE PROGRESSIVE DAMAGE ANALYSIS ON THE SUBSTRUCTURE WITH AN INITIAL FLAW

Now, since the cohesive damage mechanism is not the most important one in the previous analysis, to deepen it an initial flaw is introduced between every stiffener and the composite panel. In real life, this kind of damage can be the result of environmental degradation or bad adhesion. The goal is to understand if, in these conditions, the damage's mechanism contribution is much more significant.

Aluminium 6063AT6 is still treated like an elastoplastic material, but the damaged process is treated differently than before. Firstly, damage is now only considered in the composite panel-stiffener interface, which means that in these analyses no cohesive elements are placed in the other identified interfaces – Figure 6.11.

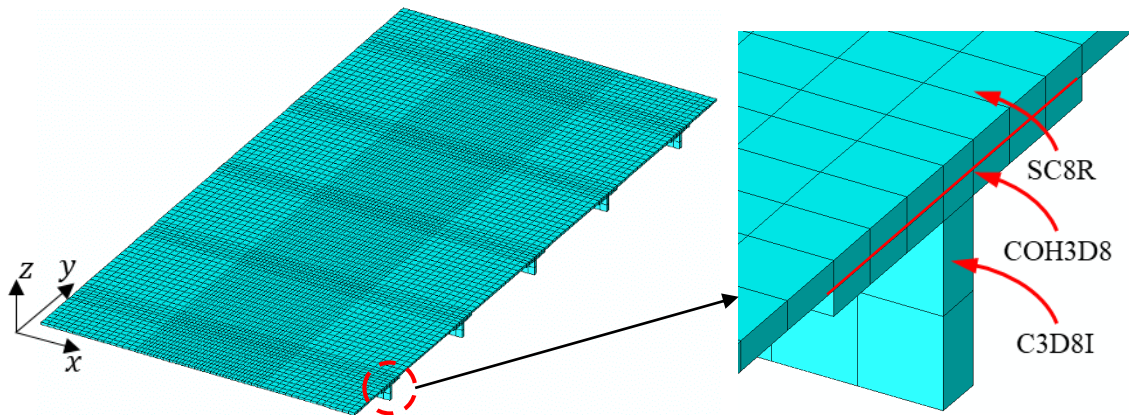


**Figure 6.11** – FE model of the substructure and used elements in the progressive damage analysis with an initial flaw.

The initial flaws are addressed by choosing an initial area – length ( $l_{in}$ )  $\times$  50 mm – in which the FE are removed in every stiffener (to simulate that the interaction between parts is compromised) and imposing contact restrictions in this area to avoid interpenetration – Figure 6.12. That contact is implemented using a surface-to-surface formulation with finite-sliding.

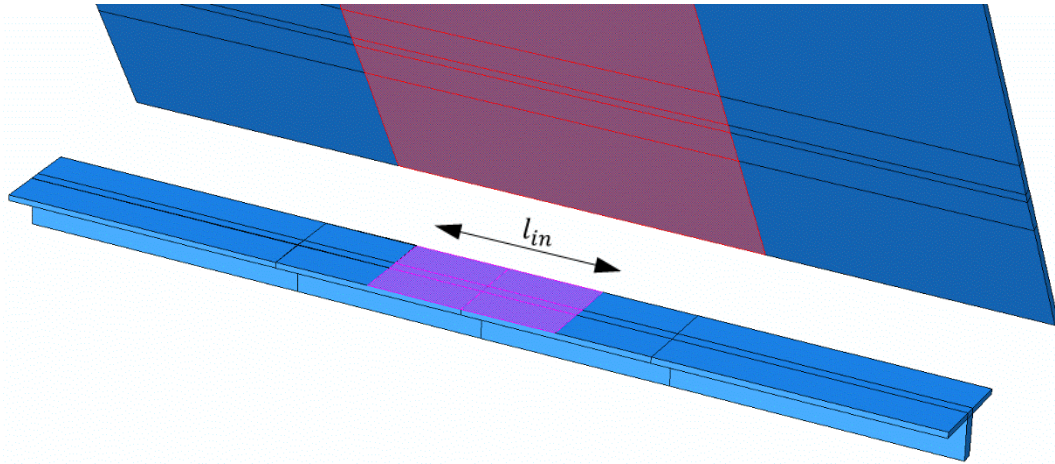


**Figure 6.12** – Contact surfaces between the composite panel and the stiffeners in the progressive damage analysis of the substructure with an initial flaw: in pink the slave surface and in red the master one. Even though only one stiffener is represented, the same occurs for all five of them.

To quantify the initial damage inflicted on the substructure, an area parameter  $\gamma$ , relating the initial damaged area with the total one of contact between the composite panel and the stiffeners, is used:

$$\gamma = \frac{\text{initial damaged area}}{\text{total area}} \times 100 = \frac{5 \times l_{in} \times 50}{5 \times 500 \times 50} \times 100 = \frac{l_{in}}{5}. \quad (6.1)$$

Boundary conditions, imposed displacements, solver, initial increment and required outputs are the same as the ones from the previous analysis and the simulation is developed for initial damages of  $\gamma = 0\%$ ,  $2\%$ ,  $10\%$  and  $20\%$ .

Figure 6.13 contains the load-displacement graphics of the simulations  $\gamma = 0\%$ ,  $2\%$ ,  $10\%$  and  $20\%$ . The same letters of Figure 6.7 are displayed.

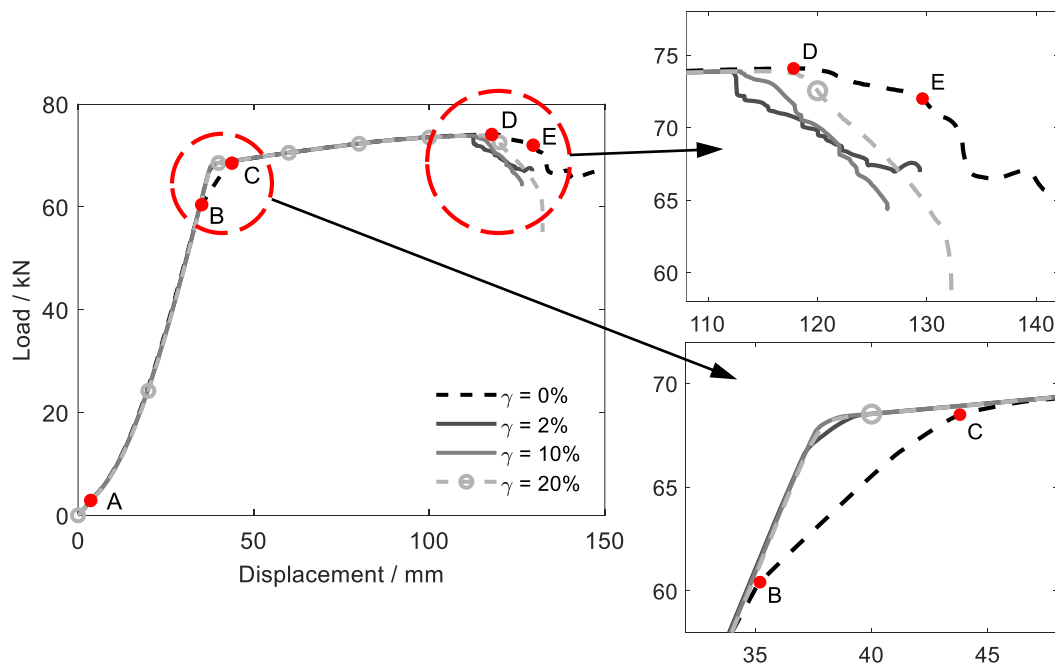


**Figure 6.13** – Load-displacement graphic of the progressive damage analysis on the substructure with different initial flaws.

Analysing its results, it is clear that, in most of the analysis, all the load-displacement curves are overlapped. Distinguishing results are found in the region where the substructure's stiffness decreases significantly (between points B and C) and when the load is dropping (near point E), as seen in the details of Figure 6.13.

To better understand what happens between points B and C, Figure 6.14 displays the central stiffener's plastic strains distribution in the moment when  $PE = 8.0\%$  is reached for every simulation. Once again, the legend used starts in  $PE = 6.0\%$  to highlight the most affected region. The images refer to displacements of 43.00 mm, 40.67 mm, 41.80 mm and 41.20 mm, respectively.

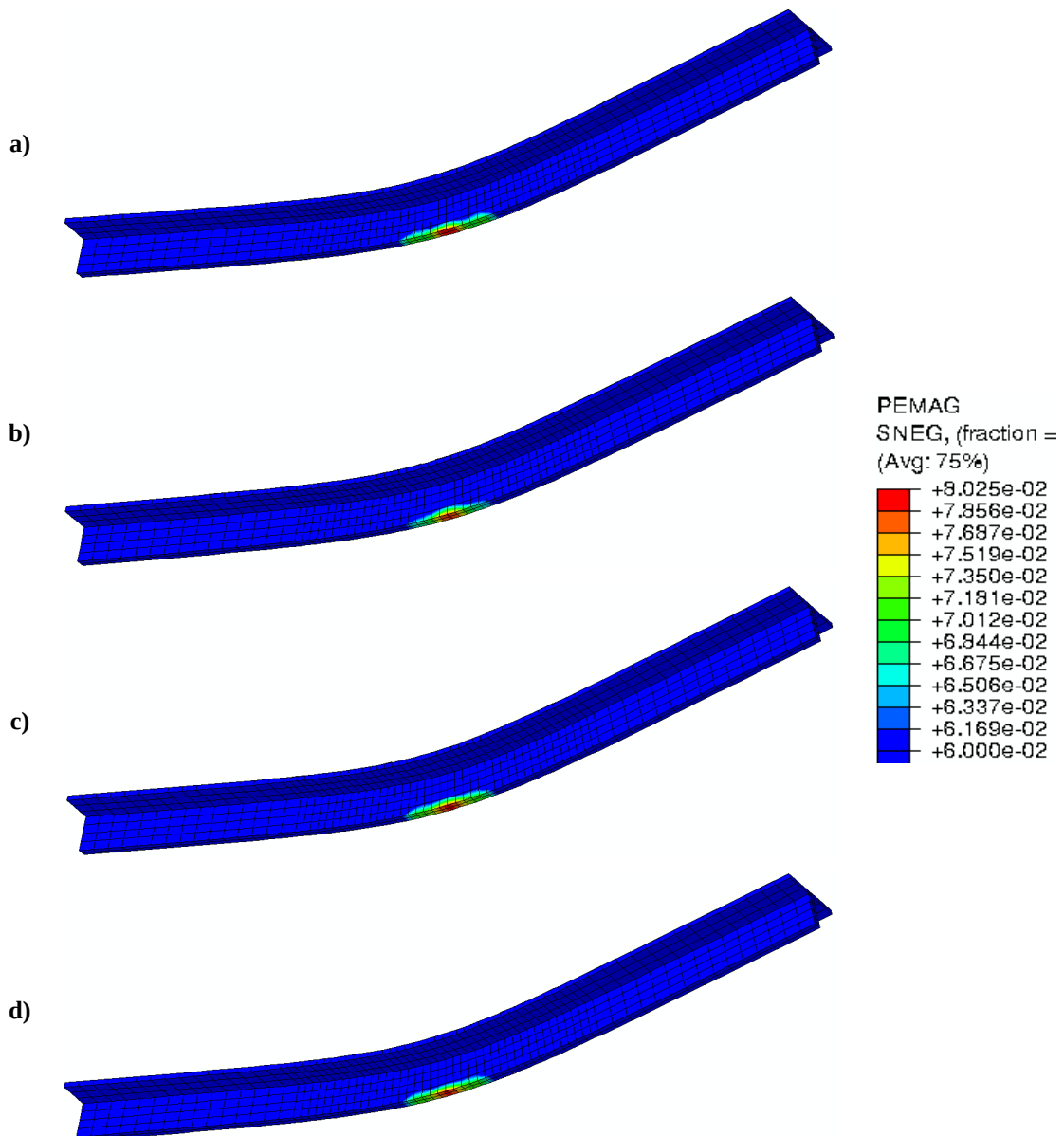


**Figure 6.14** – Plastic strain results of the progressive damage analysis on the substructure with different initial flaw sizes: (a)  $\gamma = 0\%$ ; (b)  $\gamma = 2\%$ ; (c)  $\gamma = 10\%$ ; (d)  $\gamma = 20\%$ .

Looking at these results, one sees that the distributions are very similar, but for all the simulations with an initial flaw, full plasticisation is reached earlier, for smaller displacements.

Figure 6.15 shows the damage parameter distribution for the above situations, respectively. The legend also starts in  $SDEG = 0.90$  to underline the critical regions.

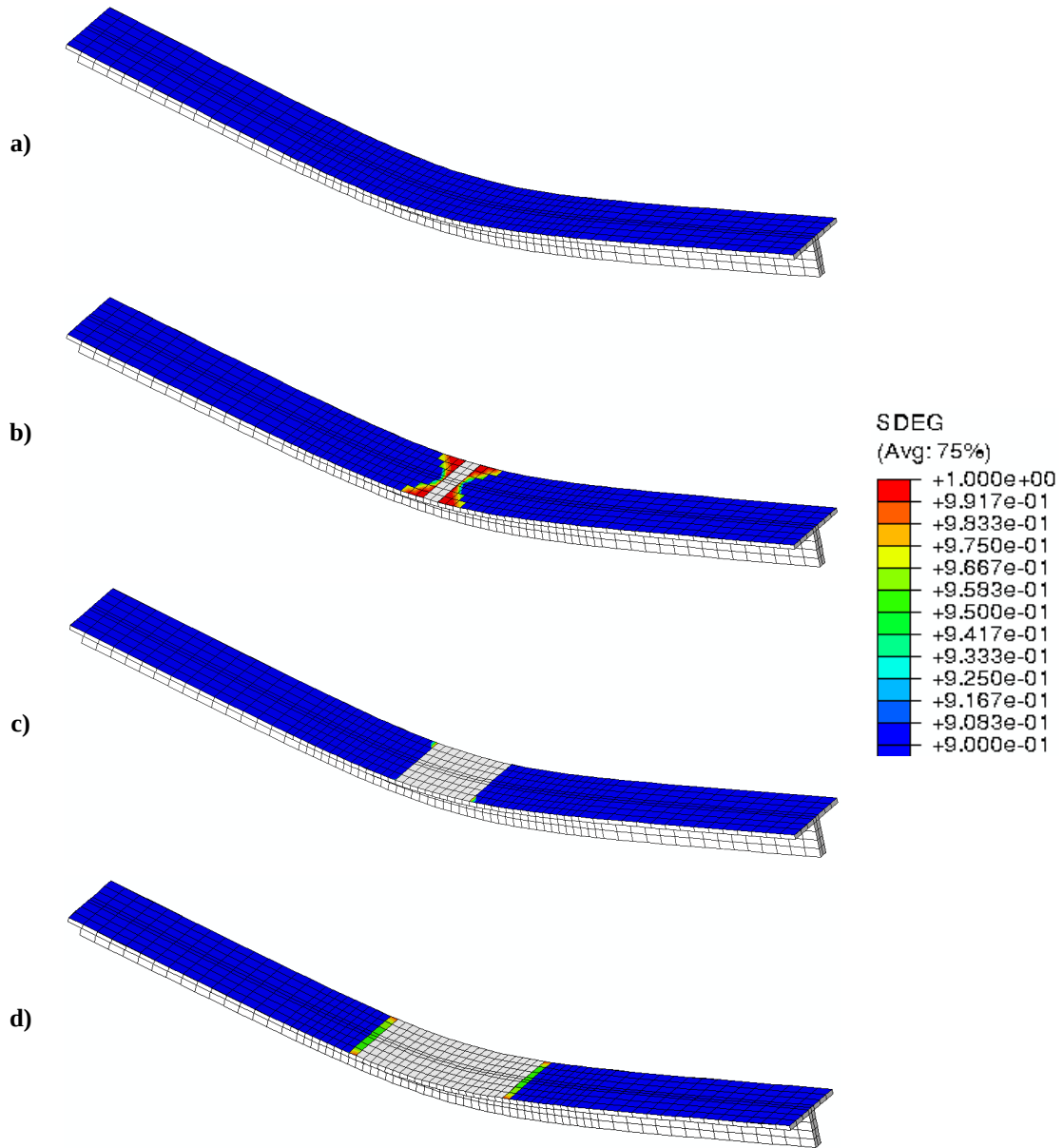


**Figure 6.15** – Damage parameter results of the progressive damage analysis on the substructure with different initial flaw sizes: (a)  $\gamma = 0 \%$ ; (b)  $\gamma = 2 \%$ ; (c)  $\gamma = 10 \%$ ; (d)  $\gamma = 20 \%$ .

In Figure 6.15 a) and Figure 6.15 c), simulations  $\gamma = 0 \%$  and  $\gamma = 10 \%$ , respectively, the damage parameter only slightly exceeds  $SDEG = 0.90$ . In Figure 6.15 d), simulation  $\gamma = 20 \%$ , the affected area is bigger than the previous ones, but the damage parameter still does not reach unity. Figure 6.15 b), simulation  $\gamma = 2 \%$ , is the one where the damage effects are more noticeable, and rupture even happens.

As intended, all the simulations with an initial flaw present more extensive damage results. In addition, common to all is the fact that the material's degradation/failure occurs in the corners of the initial flaw. These corners can be perceived as free-edges and are usually prone locations for damage

initiation. This helps to justify the fact that the  $\gamma = 2\%$  simulation is the one with the most damage effects, since its free edges from the initial flaw are in a very central region where the stresses are intensified.

Similarly to the simulation without an initial flaw, after the last point of the hardening curve the material becomes perfectly plastic, so the applied load keeps increasing. In fact, once again, the test would end shortly after the plasticisation of the stiffeners.

It is still important to point out that in this substructure the yielding of the stiffeners develops a much more important role than in the original structure. Therefore, the substructure does not perfectly represent the original structure.

Observing Table 6.2, one can compare the maximum load verified in each of the simulations. The one with the biggest applied load is the  $\gamma = 0\%$  simulation, thus the introduction of an initial defect weights in the maximum load attained. Even though the  $\gamma = 2\%$  simulation facilitates the damage process and it is the one with the lowest maximum load, the main conclusion is that the substructure's analysis is pretty much indifferent to the damage size. This probably occurs because of the testing boundaries and loading: the flaw prevents the transmission of shear efforts between the parts but, as the loading is in the downward direction, the normal efforts (which are the dominant ones in this analysis) are still transmitted. It is still important to mention, that even though in this analysis the maximum load is not modified significantly, the flaw propagated and in future scenarios, under different testing conditions, this can be decisive.

**Table 6.2** – Maximum load verified for different initial flaw sizes in the progressive damage analysis of the substructure.

|                   | $\gamma = 0\%$ | $\gamma = 2\%$ | $\gamma = 10\%$ | $\gamma = 20\%$ |
|-------------------|----------------|----------------|-----------------|-----------------|
| Maximum Load [kN] | 74.085         | 73.877         | 73.894          | 73.897          |

### 6.3. FINAL REMARKS

Throughout this chapter, multiple analyses to the chosen substructure are executed. First, a stress analysis allows to identify the most likely surfaces to suffer damage. Next, a new analysis, adding the elastoplastic behaviour of the stiffeners and cohesive elements in the critical interfaces, results in the recognition of the composite panel-stiffener interface and the debonding mechanism as the most crucial ones. From this, a final analysis like the previous one but adding multiple initial flaws in the critical interface is run. The main conclusions are that the added flaw results in an earlier plasticisation of the stiffeners and more extensive damage (with the damage initiation occurring in the free edges of the initial flaw). Also, under the current testing conditions, the maximum load attained is pretty much damage insensitive.

In the dynamic analyses of the next chapter, the identified critical damage location is used as the one where the initial damage is also placed.



## CHAPTER 7. SUBSTRUCTURE'S DYNAMIC ANALYSIS

---

### 7.1. INTRODUCTION

In this chapter, the substructure of the whole wing representative structure undergoes a dynamic analysis.

Firstly, to complete the full characterisation of the substructure's components, the damping parameters used are established – section 7.2. Then, in section 7.3, the modal analysis of the substructure is performed to know its modes of vibration. Besides considering the healthy substructure, simulations with an initial flaw are run too. Lastly, in the implicit dynamic analysis (section 7.4), after establishing the fit number of discretisation points per wave to use, the initial flaw size, the measurement location and the excitation frequency importance in the results is evaluated. This way, on section 7.5, one infers about the use of VSHM techniques in structures like the one presently studied.

### 7.2. DAMPING PARAMETERS

As mentioned in the model's formulation in section 3.5.1, in dynamic analysis, Rayleigh's damping is used. That being so, the proportional damping constants  $\alpha$  and  $\beta$ , need to be defined. The values for the USN150B composite panel are extracted from Camacho (2020). On the other hand, the ones for aluminium 6063AT6 of the stiffener are not listed and must be calculated. For that, one starts by performing the modal analysis on a stiffener with the main goal of determining its natural frequency. The imposed boundary conditions are schematised in Figure 7.1.

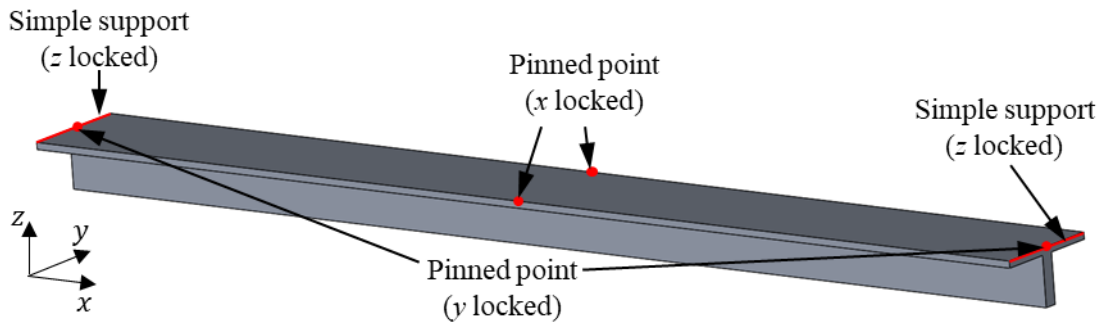


**Figure 7.1** – Boundary conditions in the modal analysis of the stiffener.

The fundamental natural frequency obtained is  $\omega_1 = 241.0 \text{ Hz} = 1514.2 \text{ rad.s}^{-1}$ . Assuming that the material viscous damping ratio is  $\xi = 0.005$  (Wolfe et al., 1995) and  $\beta = 0$  (since there is only

one experimental result), adjusting the  $\alpha$  parameter for the first mode of vibration, it is calculated by (Petyt, 2010; Burlayenko and Sadowski, 2014)

$$\alpha = 2\xi\omega_1. \quad (7.1)$$

To favour the numerical analysis, the implemented values of the proportional damping constants are doubled. The final parameters are listed in Table 7.1.

**Table 7.1** – Proportional damping parameters.

| Material          | $\alpha$ [rad. s <sup>-1</sup> ] | $\beta$ [s. rad <sup>-1</sup> ] |
|-------------------|----------------------------------|---------------------------------|
| USN150B           | 7.54                             | $6.78 \times 10^{-6}$           |
| Aluminium 6063AT6 | 30.28                            | 0                               |

### 7.3. MODAL ANALYSIS

The goal of this section is to develop a modal analysis of the chosen substructure.

The FE model used is like the ones from Chapter 6: the composite panel is discretised by SC8R elements and the stiffeners by C3D8I ones. The average element size is 5 mm and the mesh is conformal.

Besides the sane substructure, the initially damaged situation is considered here. The initial flaw is inserted in the central region of the substructure, between the composite panel and the stiffener. This is implemented by not tying this region as presented in Figure 7.2.

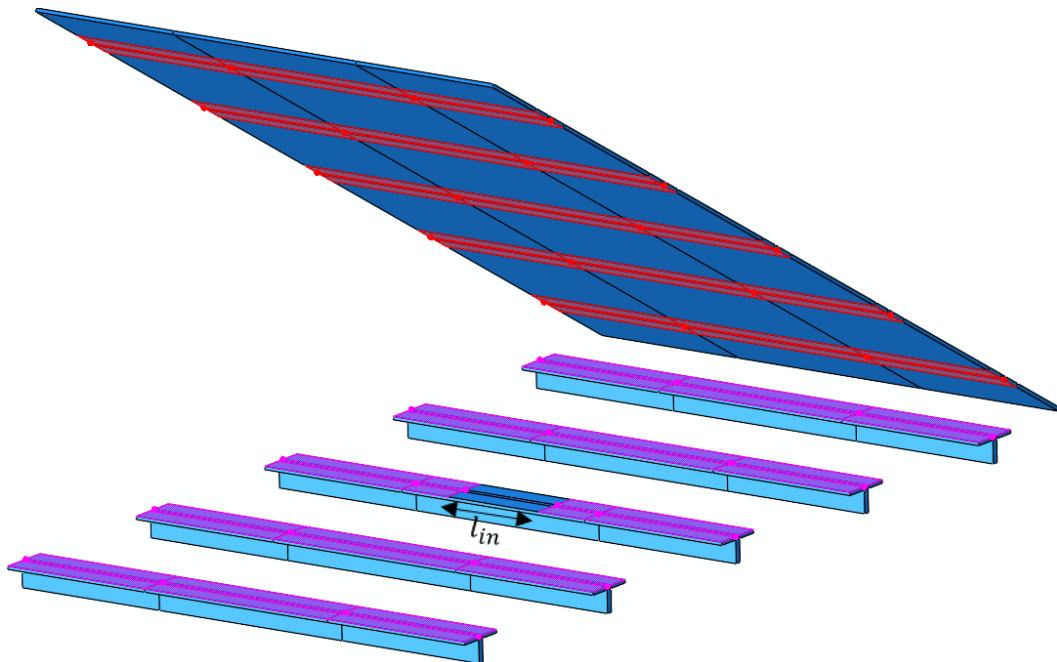


**Figure 7.2** – Tie interaction between the composite panel and the stiffeners in the modal analysis of the substructure. The central damaged region is not tied.

This flaw is quantified by

$$\gamma = \frac{\text{initial damaged area}}{\text{total area}} \times 100 = \frac{l_{in} \times 50}{5 \times 500 \times 50} \times 100 = \frac{l_{in}}{25}. \quad (7.2)$$

Once again, the software is incapable of considering any nonlinearity. Therefore, in this analysis, the software assumes that the nodes of this region are not tied and the interaction between the parts in the flawed region is not taken into account.

The substructure is submitted to the boundary conditions of Figure 7.3, where the substructure is sustained on two roller supports along its width and has pinned points along the  $x$  and  $y$  directions to avoid a rigid body motion in the  $y$  and  $x$  direction, respectively. The eigenvalue extraction method applied is Lanczos and the requested outputs are the first twenty modes of vibration.

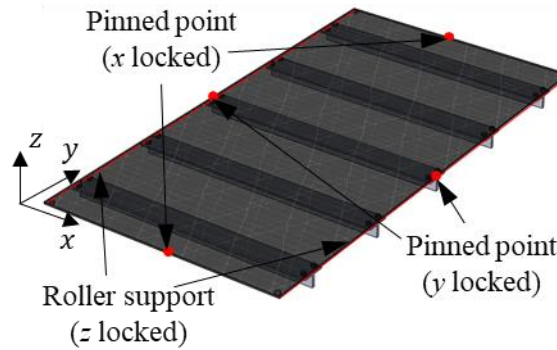


**Figure 7.3** – Boundary conditions in the modal analysis of the substructure.

Table 7.2 lists the first six natural frequencies for simulations with different initial flaws.

**Table 7.2** – First six natural frequencies obtained for different initial flaw sizes in the modal analysis of the substructure.

| $\omega_i$ [rad.s <sup>-1</sup> ] | $\gamma = 0\%$ | $\gamma = 0.4\%$ | $\gamma = 2\%$ | $\gamma = 4\%$ | $\gamma = 8\%$ |
|-----------------------------------|----------------|------------------|----------------|----------------|----------------|
| $i = 1$                           | 1298.8         | 1298.8           | 1298.8         | 1298.8         | 1298.4         |
| $i = 2$                           | 1298.9         | 1298.9           | 1298.9         | 1298.9         | 1298.6         |
| $i = 3$                           | 1357.5         | 1357.5           | 1357.2         | 1355.6         | 1333.6         |
| $i = 4$                           | 1425.2         | 1425.1           | 1425.0         | 1424.2         | 1418.6         |
| $i = 5$                           | 1567.6         | 1567.6           | 1567.5         | 1566.6         | 1556.3         |
| $i = 6$                           | 1777.0         | 1777.0           | 1777.0         | 1777.0         | 1776.9         |

Looking closely at the natural frequencies, the highest frequency is the one from the sane ( $\gamma = 0\%$ ) substructure and as the damaged area increases so does the difference between the registered frequencies. The decrease of these frequencies agrees with the loss of the substructure's stiffness by the introduction of a flaw.

Nonetheless, the flaws considered have a negligible influence on the natural frequencies of vibration; for example, the difference between the first natural frequency of the structures without flaw and with  $\gamma = 8\%$  is only 0.031 %. In engineering practice, it would be nearly impossible to detect this type of flaws from the natural frequencies.

Next, Figure 5.14 presents the respective modes of vibration for  $\gamma = 0\%$ . For the other simulations, the modes of vibration are not represented here since they resemble the modes of the sane substructure.

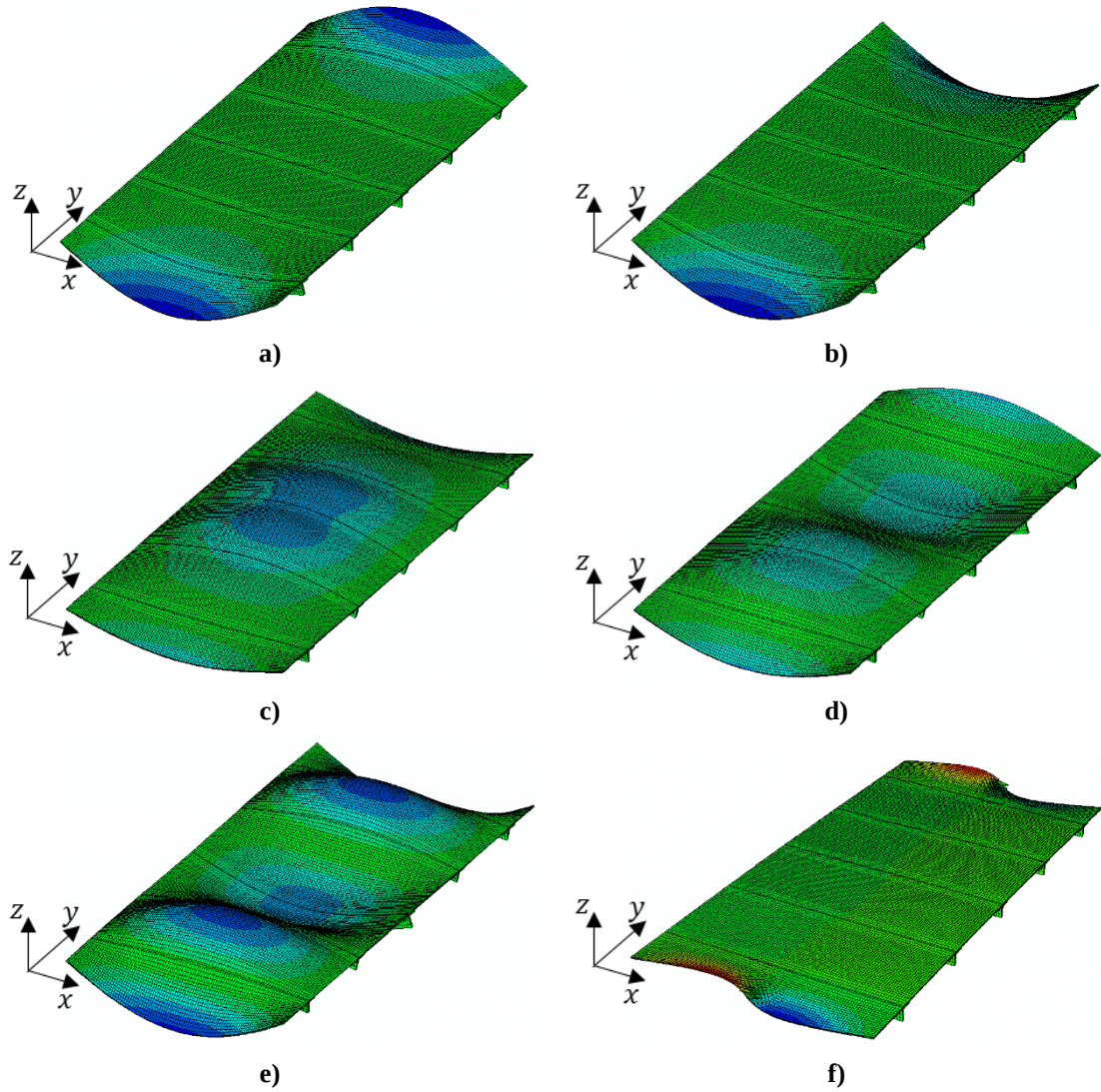


**Figure 7.4** –  $i^{th}$  mode shape of the substructure: a)  $i = 1$ ; b)  $i = 2$ ; c)  $i = 3$ ; d)  $i = 4$ ; e)  $i = 5$ ; f)  $i = 6$ .

Observing these results, one concludes that the first 5 modes of vibration are flexional ones and the 6<sup>th</sup> is a torsional one. Individually, one can mention about each mode that:

- in the 1<sup>st</sup> mode shape, the composite panel's extremities vibrate in phase opposition and the central line functions as a node;

- in the 2<sup>nd</sup> mode shape, the extremities vibrate in phase with each other but in opposition with the central region (however the amplitude in the central region is really small compared to the extremities);
- in the 3<sup>rd</sup> mode shape, the extremities are in phase but in opposition with the central region. There are two lines near the first and last reinforcements that are a node;
- in the 4<sup>th</sup> mode shape, the composite panel is divided in four regions that vibrate in phase opposition with the adjacent ones. There three lines that are a node: the central one and near the top of the first and last reinforcements;
- in the 5<sup>th</sup> mode shape, the extremities and central region vibrate in phase, but the adjacent regions are in phase opposition. There four lines that are a node and they coincide with the first, second, fourth and last reinforcement;
- in the 6<sup>th</sup> mode shape, the composite panel's free extremities twist and the remaining panel behaves like a node.

It is verified that the implemented flaws affect only slightly the natural modes of vibration, both in what concerns the frequencies and the natural mode shapes. Hence, when vibrating in one of its first six modes, the damaged structure is still able to transmit stresses and accelerations between its different regions in a way that does not differ much from the one of the sane structure.

## 7.4. IMPLICIT DYNAMIC ANALYSIS

In order to execute the desired implicit dynamic analysis of this section, first of all, a simple convergence test with the healthy substructure is carried out to determine the time increment to use hereafter.

Then, multiple dynamic analyses are run and the influence of the initial flaw size, the measurement location and the excitation frequency are explored in sections 7.4.2, 7.4.3 and 7.4.4, respectively.

### 7.4.1. CONVERGENCE STUDY OF THE DYNAMIC ANALYSIS ON THE SUBSTRUCTURE

The present convergence test intends to determine the fit value for the time increment  $\Delta t$  used in the time discretisation process of the dynamic analysis on the healthy substructure.

Like presented in the mathematical model, the type of dynamic analyses implemented is an implicit dynamic one with a duration of 2 s. Also, the material properties considered are only elastic ones and geometrical nonlinearities are neglected.

The boundary conditions of the problem are the exact ones from the previous modal analysis. On the other hand, there is now an applied harmonic load in the central region of the composite panel – Figure 7.5.

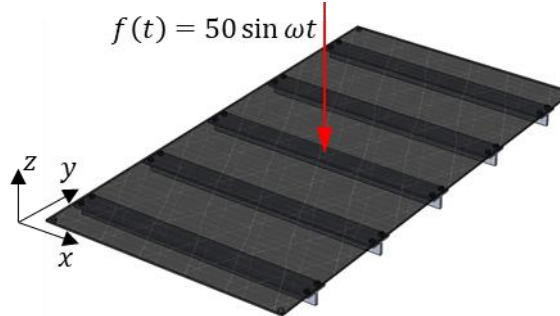


**Figure 7.5** – Harmonic loading applied on the central region in the substructure for the dynamic analysis.

The implemented FE model is the same from the progressive damage analyses, but the tie constraint is enforced in all the composite panel-stiffener interface (there is no initial damage).

The time discretisation is then performed by the HHT time integrator with the parameters of Table 7.3 and, to solve the nonlinear problem resultant from the addition of contact to the problem, Newton's method is used. In the ABAQUS software, these parameters are suggested for contact situations that benefit from energy dissipation to help improving convergence without significantly degrading the solution's accuracy.

**Table 7.3** – Parameters for the HHT integrator used in the dynamic analysis of the substructure.

| $\alpha_{HHT}$ | $\beta_N$ | $\gamma_N$ |
|----------------|-----------|------------|
| -0.41421       | 0.5       | 0.91421    |

Finally, in these analyses the requested outputs are the displacement and velocity ( $V$ ) in the same top central point of the composite panel where the load is applied. This output is registered in a single point location as in real life the experimental measures would be of this kind.

Therefore, next, Table 7.4 presents the developed convergence test for  $\omega = 500 \text{ rad.s}^{-1}$ . This frequency is not related in any way to the natural ones of the substructure since near them convergence can be harder to reach.

To choose the time increments used, it is common to think about the number of points needed to discretise properly one excitation wave (in this case 20, 40 and 120 points per wave) and apply to them the following equation:

$$\Delta t = \frac{2\pi}{\omega \cdot nr \text{ points}}. \quad (7.3)$$

**Table 7.4** – Time increments used, and time spent for different number of discretisation points per wave at  $\omega = 500 \text{ rad.s}^{-1}$  in the dynamic analysis of the substructure.

| Points per Wave | $\Delta t$ [s]           | Time        |
|-----------------|--------------------------|-------------|
| 120             | $1.04720 \times 10^{-4}$ | 19 h 35 min |
| 40              | $3.14159 \times 10^{-4}$ | 6 h 58 min  |
| 20              | $6.28319 \times 10^{-4}$ | 3 h 21 min  |

Figure 7.6 and Figure 7.7 display the time-history and the phase portrait, respectively, for the simulations with different time increments at  $\omega = 500 \text{ rad.s}^{-1}$ .

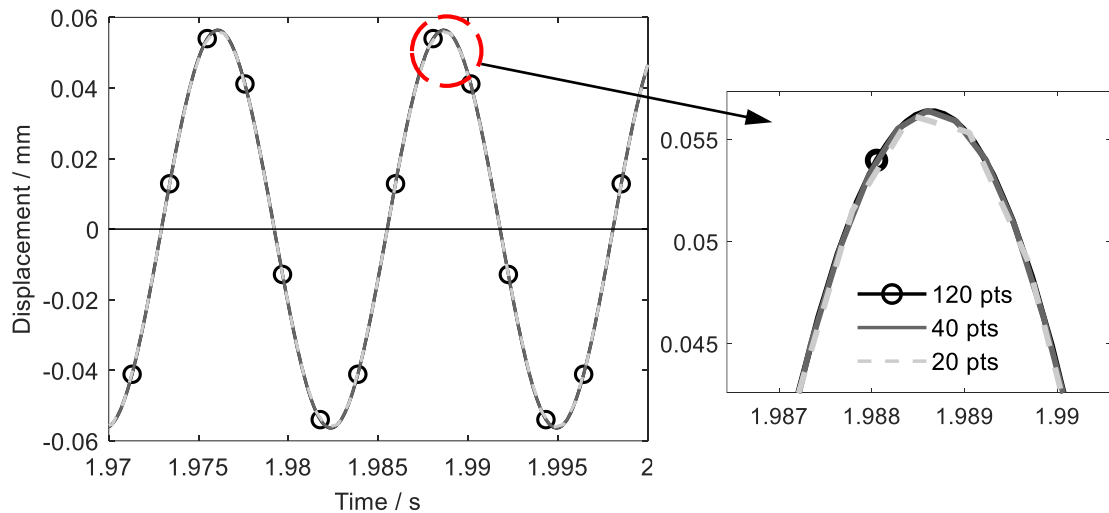


**Figure 7.6** – Time-history of the dynamic converge study on the substructure, for a different number of discretisation points per wave, at  $\omega = 500 \text{ rad.s}^{-1}$ .

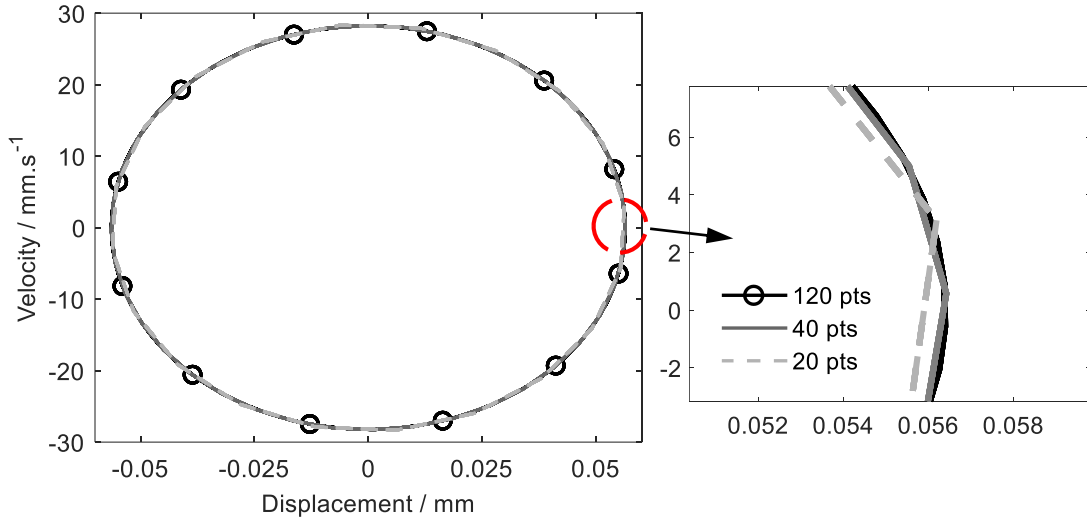


**Figure 7.7** – Phase portrait of the dynamic converge study on the substructure, for a different number of discretisation points per wave, at  $\omega = 500 \text{ rad.s}^{-1}$ .

From these results, it is possible to deduce that the use of more discretisation points per wave, and, consequently, smaller time increments, results in a bigger time spent to perform the simulation.

Also, when a higher number of discretisation points is applied, the graphics resemble more accurately a perfect harmonic wave, in the time-history, and an ellipse, in the phase portrait. In the other results, especially for the simulation with 20 discretisation points, these shapes are “discretised” more roughly.

Looking closely to Figure 7.6 and Figure 7.7 and pondering with the time spent per simulation, the most suitable number of discretisation points is 40. Its results are very similar to the ones of the simulation with 120 discretisation points but with the advantage of needing less than a third of the time. This time reduction is very valuable right now and its importance only keeps increasing when dealing with more complicated problems, like the ones from the next chapters, where nonlinearities are introduced.

Finally, note that, since the lines in the phase portraits describe a closed trajectory and overlap every cycle resulting in a single line for each simulation, the duration of the simulation is enough, and the substructure’s dynamic response has reached the stationary stage.

#### 7.4.2. NUMERICAL RESULTS OF THE DYNAMIC ANALYSIS ON THE SUBSTRUCTURE – INITIAL DAMAGE SIZE INFLUENCE

This section arises with the main goal of better understanding the influence of the initial flaw size in the numerical results obtained.

To do so, multiple simulations with different initial flaws, characterised by the parameter of Equation (7.3), are run:  $\gamma = 0.4 \%$ ,  $\gamma = 2 \%$ ,  $\gamma = 4 \%$  and  $\gamma = 8 \%$ . The boundary conditions, loads, and solver are the same as the ones from section 7.4.1.

The FE model is also like the one from the previous analysis, but contact is added to the composite panel-central stiffener interface when an initial flaw is introduced – Figure 7.8. This way, the simulation mimics a flaw between the mentioned components and allows their interaction, but not interpenetration.

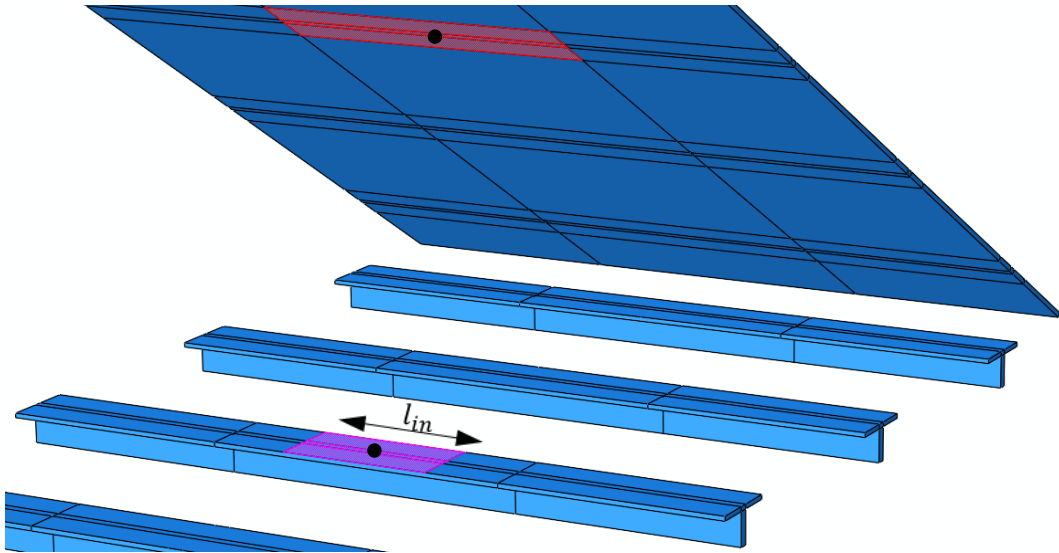


**Figure 7.8** – Contact surface between the composite panel and the stiffener in the dynamic analysis on the substructure: in pink the slave surface and in red the master one.

In the simulations with flaw, the output requests are still the displacements and velocities in the top central point of the composite panel but, additionally, these outputs are requested in two points in the centre of the contact too. These points are schematised with black dots in Figure 7.8.

The time discretisation methodology is the same too. However, the number of points used in each simulation to discretise the excitation wave is now bigger: 120 points for  $\gamma = 0.4 \%$ , 160 points for  $\gamma = 2 \%$ , 200 points for  $\gamma = 4 \%$  and 240 points for  $\gamma = 8 \%$ . One does this since the addition of contact introduces nonlinearities in the response of the system and higher-order harmonics may appear on it.

Figure 7.9, Figure 7.10, Figure 7.11, and Figure 7.12 display, respectively, the time-histories, phase portraits, frequency spectra and Poincaré sections of the developed simulations in the top central point of the composite panel. In each of these, the results are overlapped with the ones from the analysis to the sane substructure to simplify the comparison.

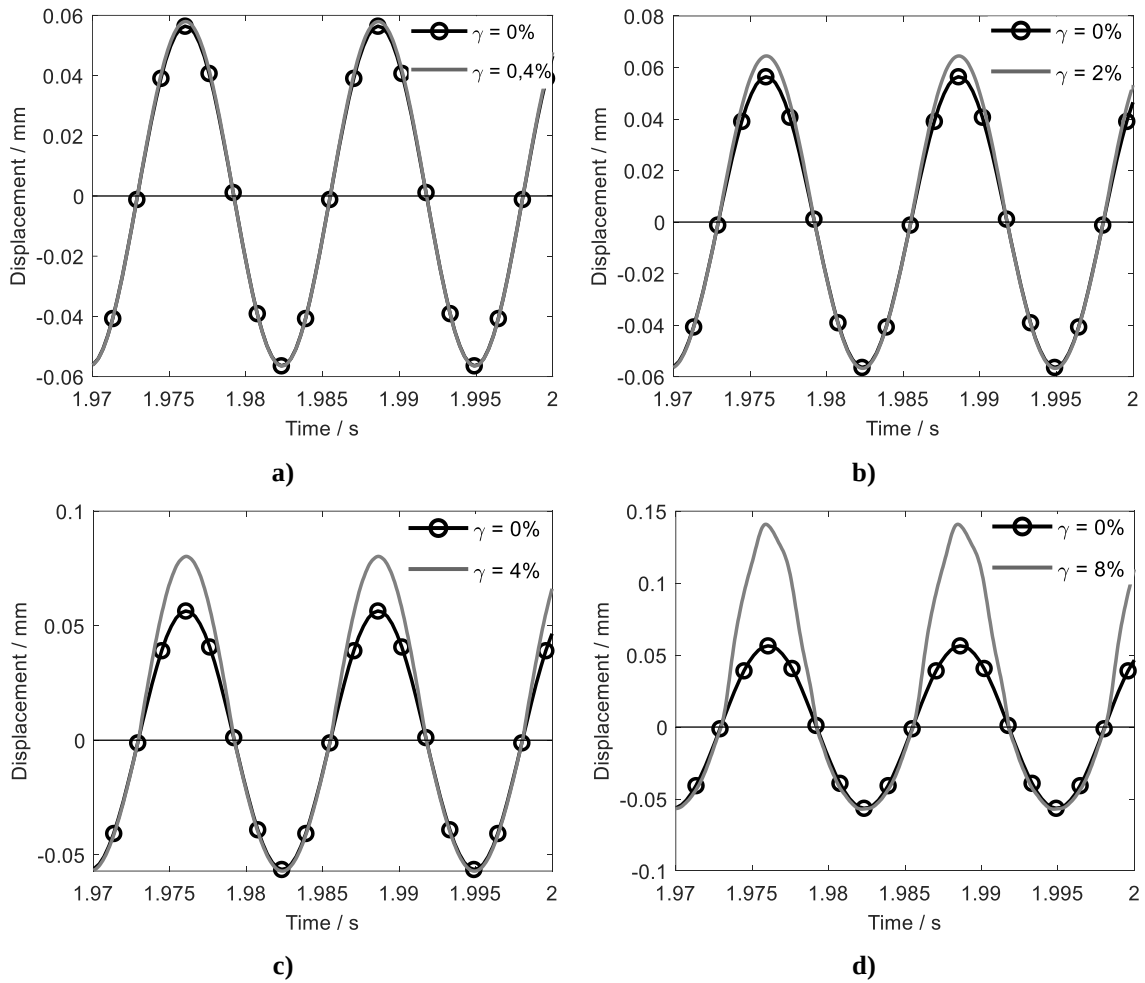


**Figure 7.9** – Time-history of dynamic analysis on the substructure at  $\omega = 500 \text{ rad.s}^{-1}$  with initial flaws of  $\gamma = 0\%$  and: a)  $\gamma = 0.4\%$ ; b)  $\gamma = 2\%$ ; c)  $\gamma = 4\%$ ; d)  $\gamma = 8\%$ .

Regarding the time-histories, first, it is important to highlight that the one from the sane substructure is a mono-harmonic response. Just like discussed in section 2.3.2, this indicates that the obtained response is linear, which is expected since no nonlinearity is introduced in this simulation.

Looking now to the time-histories of the simulations with an initial flaw, several conclusions may be drawn:

- the fundamental period of the responses remains the one of the original simulation;
- in all the analyses, the acquired results are asymmetrical (in relation to the static position), and it is possible to observe that in the part of the cycle where displacements are negative, there is contact between the plate and the stiffener, as if the flaw was not there: the behaviour of the damaged and undamaged structure is similar. When displacements are positive, the plate and the stiffener separate, hence the displacement achieves larger magnitudes in the larger than in the sane structure. This results in not mono-harmonic responses;
- the greater the initial flaw introduced, the greater the difference between the responses obtained and the more nonlinearities observed – for the smaller damaged areas it is only possible to perceive the asymmetry of the response wave but for bigger damaged areas it is also noticeable that the wave loses its “sinusoidal shape” and becomes asymmetric vertically.

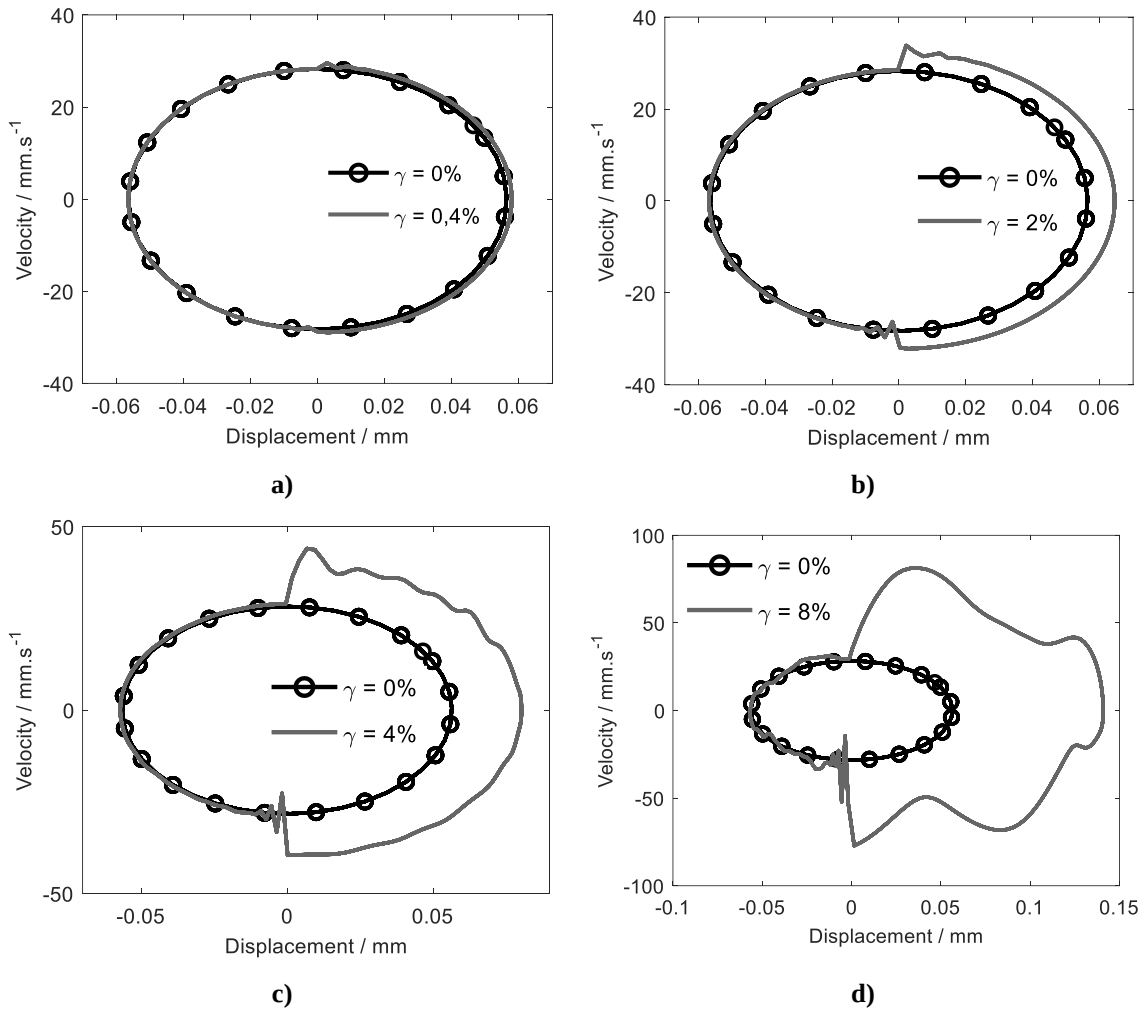


**Figure 7.10** – Phase portraits of the dynamic analysis on the substructure at  $\omega = 500 \text{ rad.s}^{-1}$  with initial flaws of  $\gamma = 0\%$  and: a)  $\gamma = 0.4\%$ ; b)  $\gamma = 2\%$ ; c)  $\gamma = 4\%$ ; d)  $\gamma = 8\%$ .

Now, in the phase portraits, the outcome from the sane substructure simulation is a perfect ellipse origin centred. Once again, this is characteristic of linear responses.

With the introduction of the initial damage, the biggest differences in the phase portrait occur, again, for positive displacements. Just like mentioned for the time-histories, this may be explained by the direction of the loading and the discrepant stiffnesses of the components.

Regarding the importance of the initial flaw size, it is observable that for smaller initial damaged areas the difference to the sane results is more discreet and there is almost symmetry with respect to the horizontal axis defined by null velocity. For bigger damaged areas, the phase portrait becomes more complex, and no symmetry can be identified.

It is still observable in all phase portraits, but mostly on the ones from simulations  $\gamma = 4\%$  and  $\gamma = 8\%$ , that in the south region there is an abrupt variation of the velocity. This is consistent with the impact of the detached parts when they are reattached.

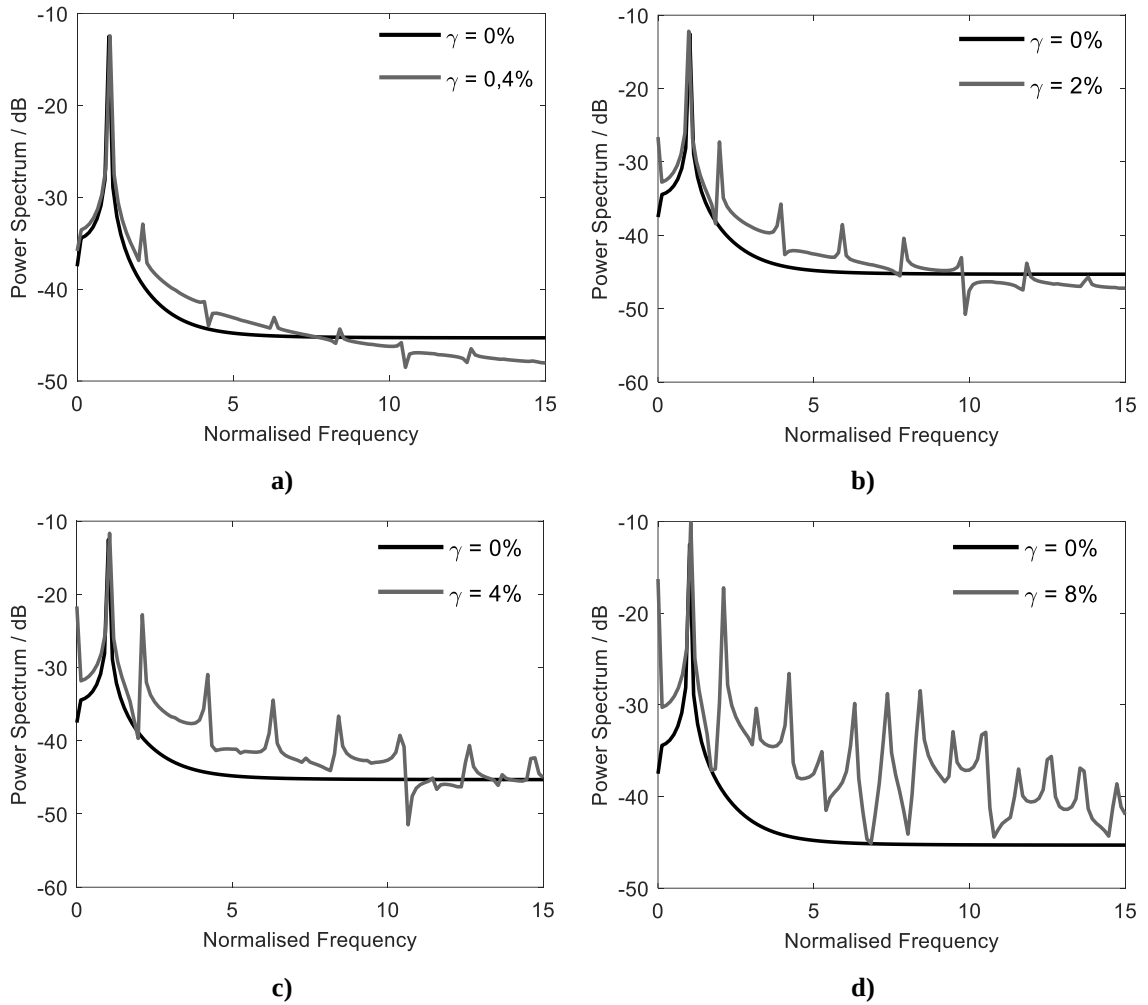


**Figure 7.11** – Frequency spectrum of the dynamic analysis on the substructure at  $\omega = 500 \text{ rad.s}^{-1}$  with initial flaws of  $\gamma = 0 \%$  and: a)  $\gamma = 0.4 \%$ ; b)  $\gamma = 2 \%$ ; c)  $\gamma = 4 \%$ ; d)  $\gamma = 8 \%$ .

To obtain the frequency spectrum, the time-histories calculated are processed by using the Fast Fourier Transform (FFT) within the MATLAB® software environment to convert them into frequency domain data. The presented frequency spectra have a normalised frequency axis, which is achieved by dividing the frequency results by the excitation frequency  $\omega = 500 \text{ rad.s}^{-1}$ .

The results of the undamaged substructure's simulation confirm that the dominant harmonic is the fundamental one.

Likewise, in the other simulations with an initial damage this harmonic is the most important one. However, when the initial flawed area increases, so does the contribution of higher-order harmonics, that are multiples of the excitation frequency. These harmonics are the result from the periodically colliding of the detached surfaces.

In the  $\gamma = 0.4 \%$  simulation, even though there is a bigger contribution from higher-order terms than in the sane response, these are still pretty much negligible. In the  $\gamma = 2 \%$  and  $\gamma = 4 \%$  simulations, even higher-order harmonics start to stand out, especially the 2<sup>nd</sup> one. Finally, in the  $\gamma = 8 \%$  simulation, the contribution of the 2<sup>nd</sup> and 4<sup>th</sup> harmonics is highlighted. The appearance of even harmonics in the response is consistent with the lack of symmetry in the phase plots.

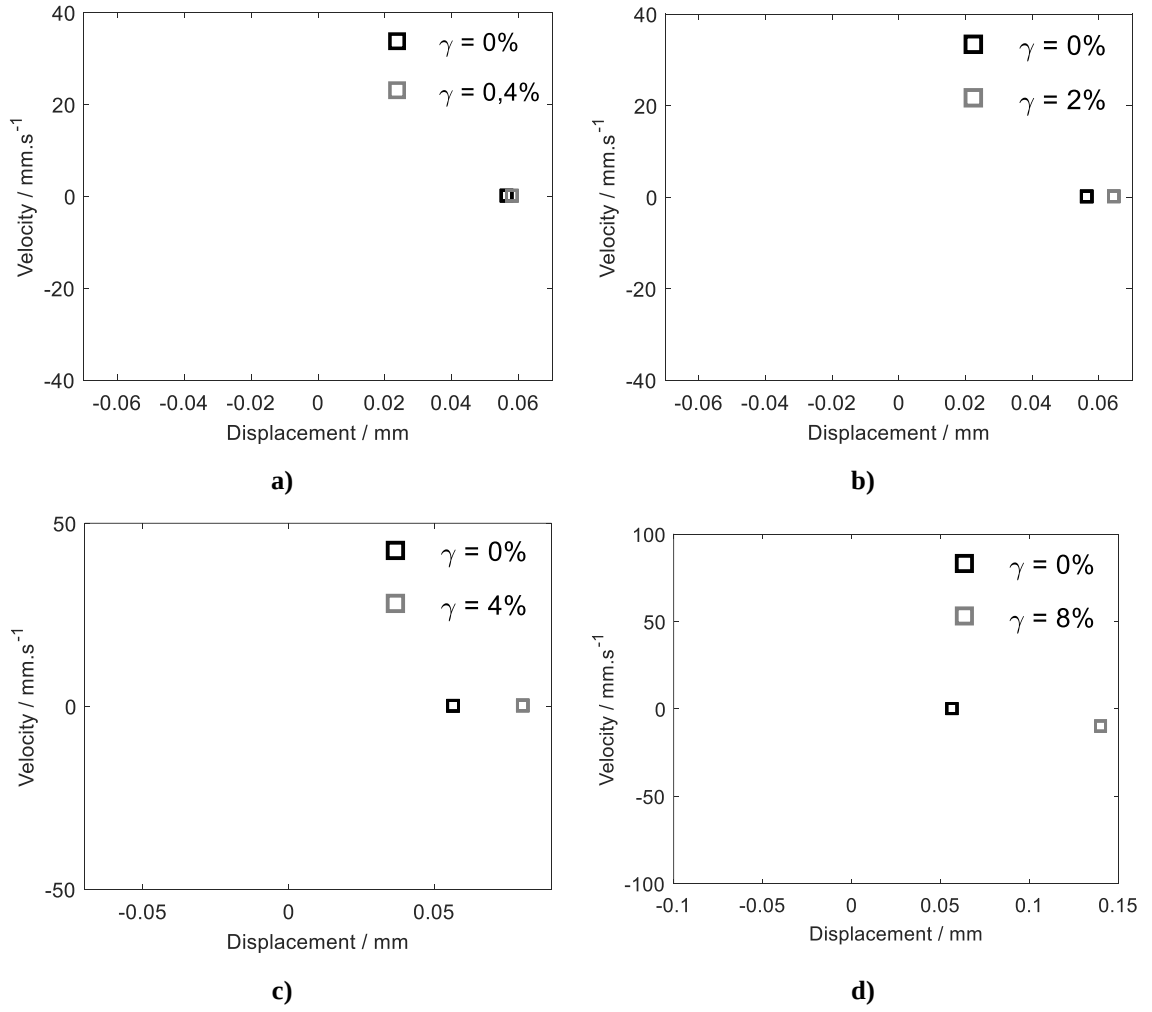


**Figure 7.12** – Poincaré sections of the dynamic analysis on the substructure at  $\omega = 500 \text{ rad.s}^{-1}$  with initial flaws of  $\gamma = 0\%$  and: a)  $\gamma = 0.4\%$ ; b)  $\gamma = 2\%$ ; c)  $\gamma = 4\%$ ; d)  $\gamma = 8\%$ .

Lastly, the Poincaré sections from Figure 7.12 are constructed by registering points separated by a period  $T$  such as (Nayfeh and Balachandran, 1994)

$$T = \frac{2\pi}{\omega}. \quad (7.4)$$

To see more distinctive results, the points are chosen in a way that guarantees that these points are near the maximum positive displacement. The axes from these images are the same as the ones from Figure 7.10.

Furthermore, the periodicity of all the responses is one as only one point is visible in the images. In the results of the initially damaged simulations, it is possible to verify that for bigger flaws, the Poincaré section shows a bigger difference between the registered points, and it is easier to identify the existence of a nonlinearity.

After analysing, in detail, the four tools above, it is possible to state some overall conclusions of this section:

- even though, the used methodologies are all different and represent the substructure's response in different ways, the conclusions attained from their analysis are accordant and complementary, which reassures them;
- in the results of the simulations with an initial flaw imposed, the greater the flaw, the greater the effect on the results when comparing them with the ones from the sane substructure.

For really small damage situations, like the one from simulation  $\gamma = 0.4 \%$ , it is still possible to verify numerically differences between the results. The variations are really small in terms of displacement but are significantly larger in terms of velocities or accelerations, so it is still possible to detect them with an accelerometer. This is the most interesting situation since the sooner the flaw is detected, the better.

For curiosity's sake, Figure 7.13 displays the time-history (for an initial damage of  $\gamma = 8 \%$ ) of the two extra points presented in Figure 7.8. From it, one confirms that the contact constraints are properly implemented and can observe the interaction between the detached parts. First, the detached parts come into contact once every excitation period and remain in contact most of the time. Second, the detachment between the parts occurs when their displacement is positive, which is consistent with the previously presented theory of the discrepant stiffnesses: the composite plate has a much lower stiffness and is, therefore, much more flexible, while the stiffener component remains rigid. When the displacement becomes negative, the components continue connected because the rigid stiffener limits the displacement of the composite panel.

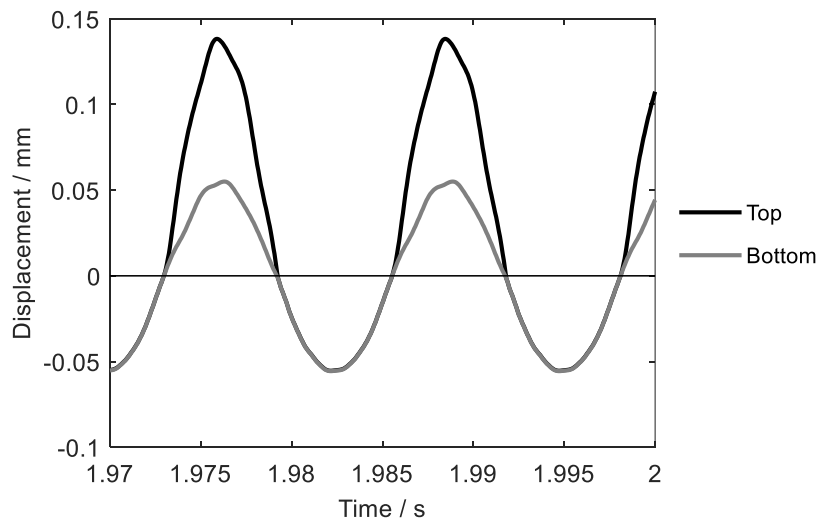


**Figure 7.13** – Time-history of the detached parts on the dynamic analysis at  $\omega = 500 \text{ rad. s}^{-1}$  with an initial flaw of  $\gamma = 8 \%$ .

### 7.4.3. NUMERICAL RESULTS OF THE DYNAMIC ANALYSIS ON THE SUBSTRUCTURE – MEASUREMENT LOCATION INFLUENCE

In this section, the previous analysis with an initial flaw of  $\gamma = 8\%$  is replicated, but now the requested outputs are registered at twelve distinct points, like shown in Figure 7.14. The location of the points is chosen to see if there is any difference in the obtained results: six of these points are in the symmetry line and other six are 75 mm to the side, and some are located above the stiffeners and others are between them.

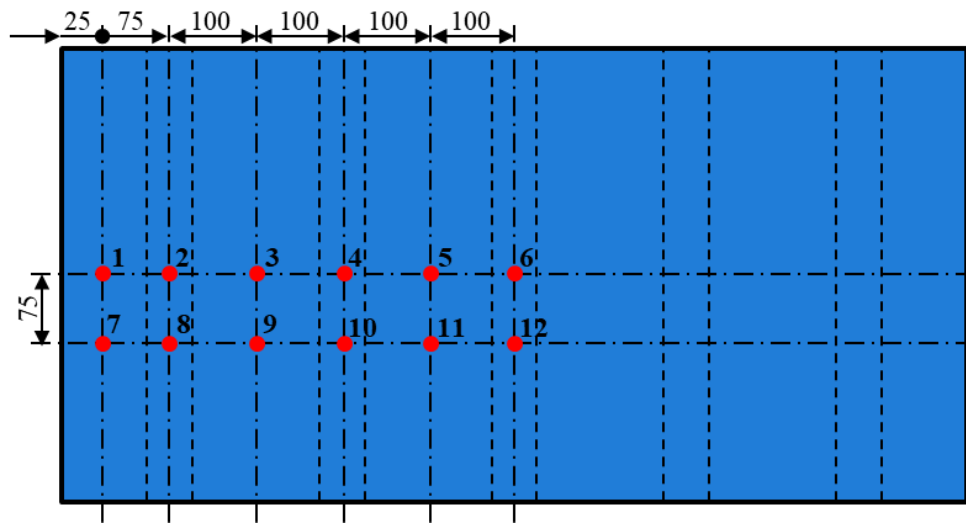


**Figure 7.14** – Location of the twelve measurement points in the substructure for the dynamic analysis.

In Figure 7.15 and Figure 7.16, the time-history of each of the twelve points is represented. The legend of these images is the one from Figure 7.15 b) and Figure 7.16 b).

From these figures, some important observations can be highlighted:

- in all time-histories, the presence of a nonlinearity (damage) is easily detected since none of the responses is a mono-harmonic one;
- in the response from the undamaged substructure, the points further away from the load point have lower amplitude responses. Consequently, the responses from the initially damaged structure also have lower amplitudes in these points;
- comparing the responses from the points in the symmetry line with the ones from the points in the 75 mm offsetted line, the responses are very similar. The differences observed are mostly due to the bigger distance to the load point, which results in slightly lower amplitudes;
- regarding the recognition of the damage locus based on the response of multiple points, in this scenario, it does not seem possible since there is a major influence in the responses of the distance to the loading point. Thus, it becomes difficult to identify if the response changes are indeed significant or small. Ideally, to eliminate this, the load should be distributed through the whole composite panel surface.

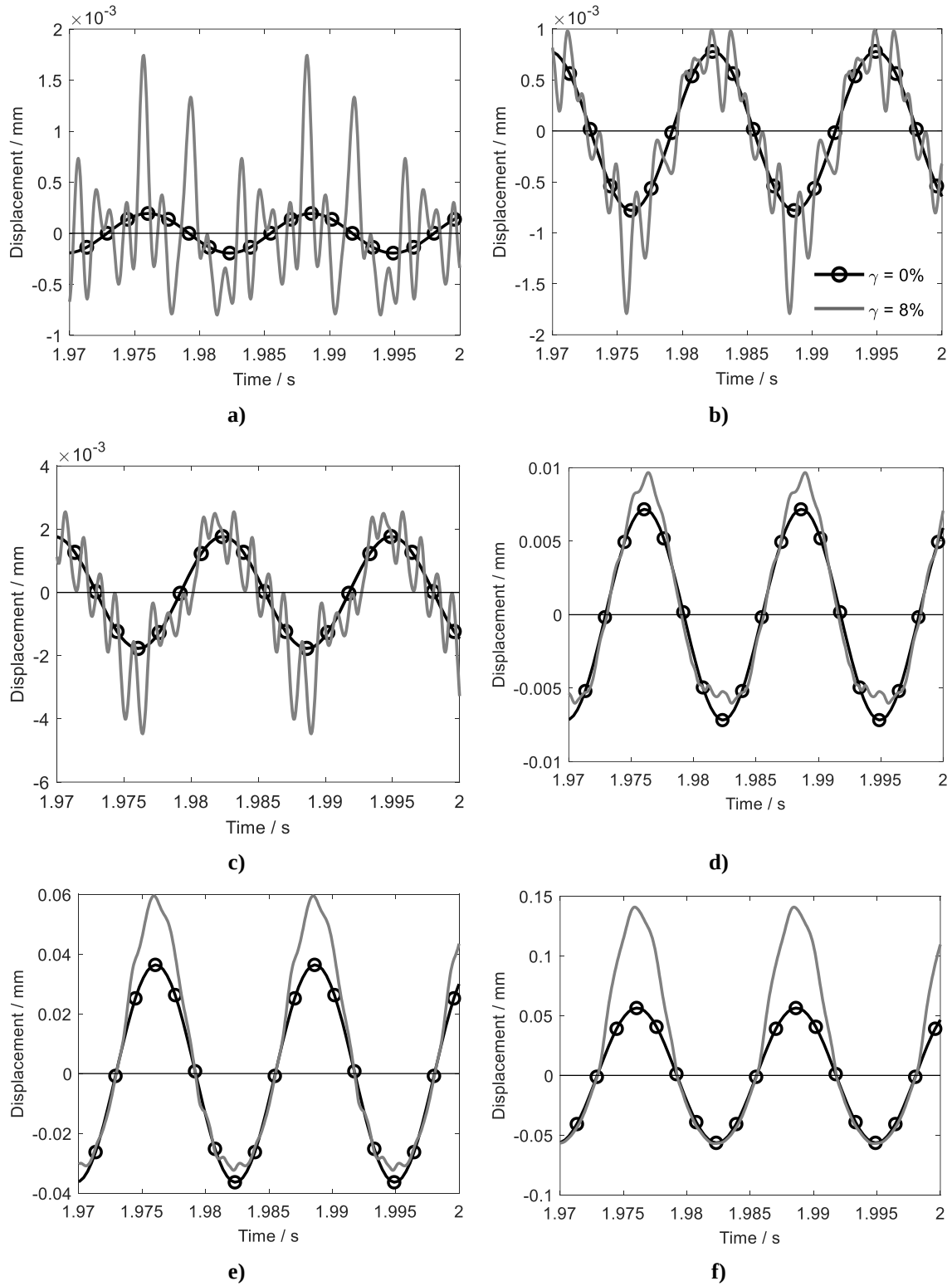


**Figure 7.15** – Time-history of the dynamic analysis on the substructure with initial flaws of  $\gamma = 0\%$  and  $\gamma = 8\%$  at  $\omega = 500 \text{ rad.s}^{-1}$  in point: a) 1; b) 2; c) 3; d) 4; e) 5; f) 6.

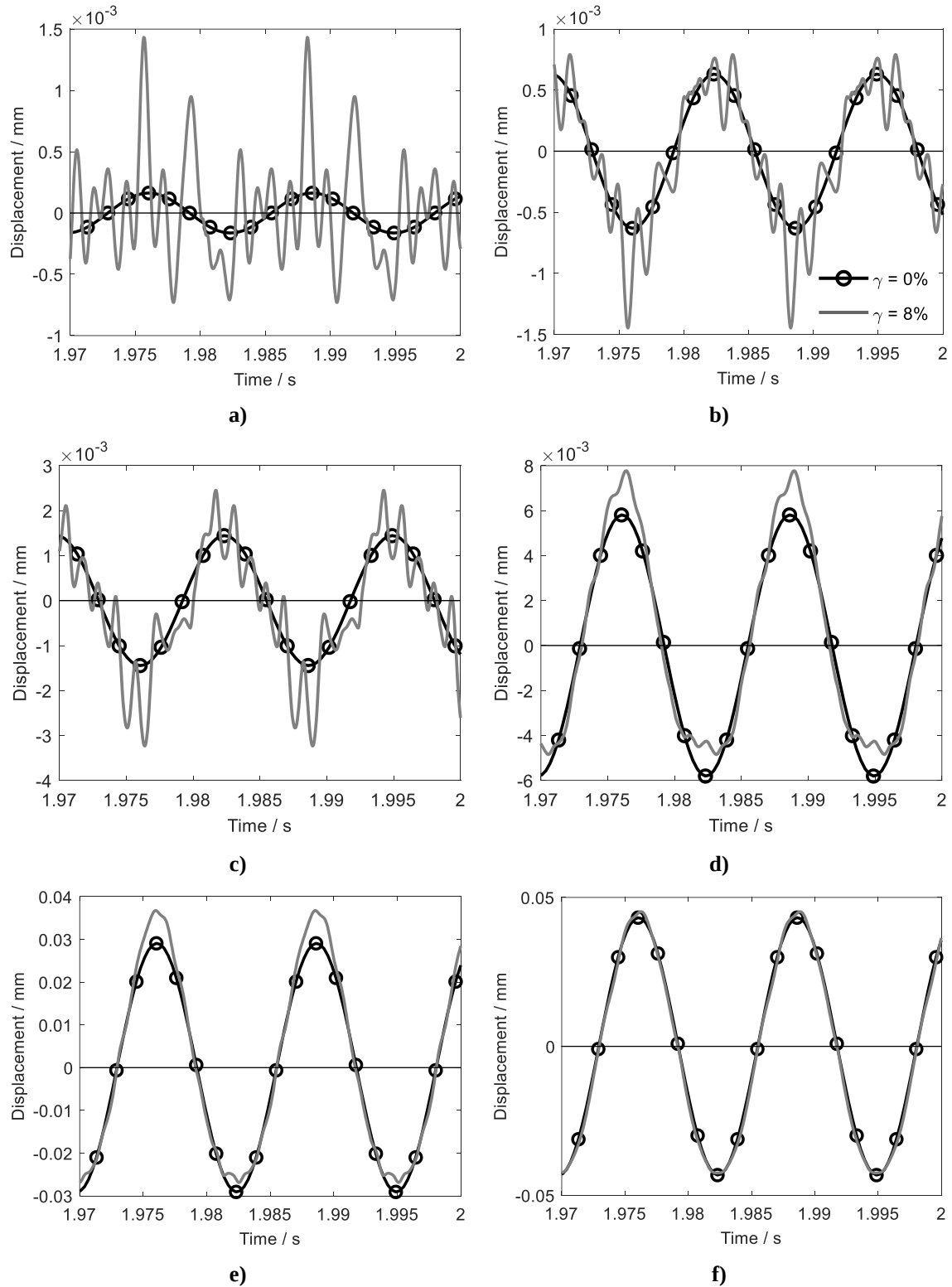


**Figure 7.16** – Time-history of the dynamic analysis on the substructure with initial flaws of  $\gamma = 0\%$  and  $\gamma = 8\%$  at  $\omega = 500 \text{ rad.s}^{-1}$  in point: a) 7; b) 8; c) 9; d) 10; e) 11; f) 12.

Additionally, looking at the responses of points 1, 2, 3, 7, 8 and 9 (all of them located closer to the composite panel's free extremity and away from the loading point) in these the contribution of multiple higher-order harmonics is clear. To better see this contribution in these responses and in the ones from the central points, Figure 7.17 contains the frequency spectrum of point 1 and 4.

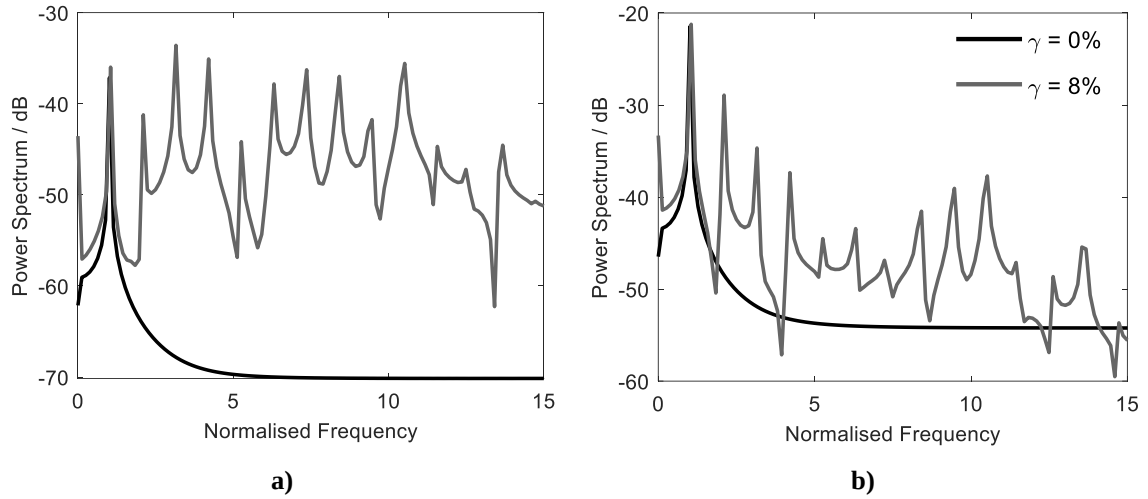


**Figure 7.17** – Frequency spectrum of the dynamic analysis on the substructure with initial flaws of  $\gamma = 0\%$  and  $\gamma = 8\%$  at  $\omega = 500 \text{ rad.s}^{-1}$  in point: a) 1; b) 4.

Analysing it (and revisiting Figure 7.11 d)), one concludes that all the responses contain contributions from higher-order harmonics of at least  $-40 \text{ dB}$ , probably provoked by the interaction between the detached parts. As in the case of points 1, 2, 3, 7, 8 and 9 the contribution from the fundamental harmonic is so small, around  $-40 \text{ dB}$  too, the higher-order harmonics stand out in their response. Also, in points 1 and 7, the local stiffness is so low that no energy from higher harmonics is filtered.

Finally, Figure 7.18 displays the Poincaré sections from points 1 to 6.

Once again, in all the graphics it is visible the presence of nonlinearities that influence the substructure's response.

Likewise, in all of them, even though the analysed damage situation is already severe, the substructure's response periodicity does not change, and it is still 1, as the single point per simulation in each image show.

In short, the only true conclusion about the importance of the measure location in the results attained is that the results from points 1 to 6 do not differ significantly from the ones from points 7 to 12, respectively.

If this influence was very well defined, it would be possible to determine based on it the unknown damage location. However, in this example, the use of a punctual load and the local variations of stiffness impact the results. The influence of the load could possibly be eliminated if a damage parameter that established the relative difference between the sane and the damaged was used.

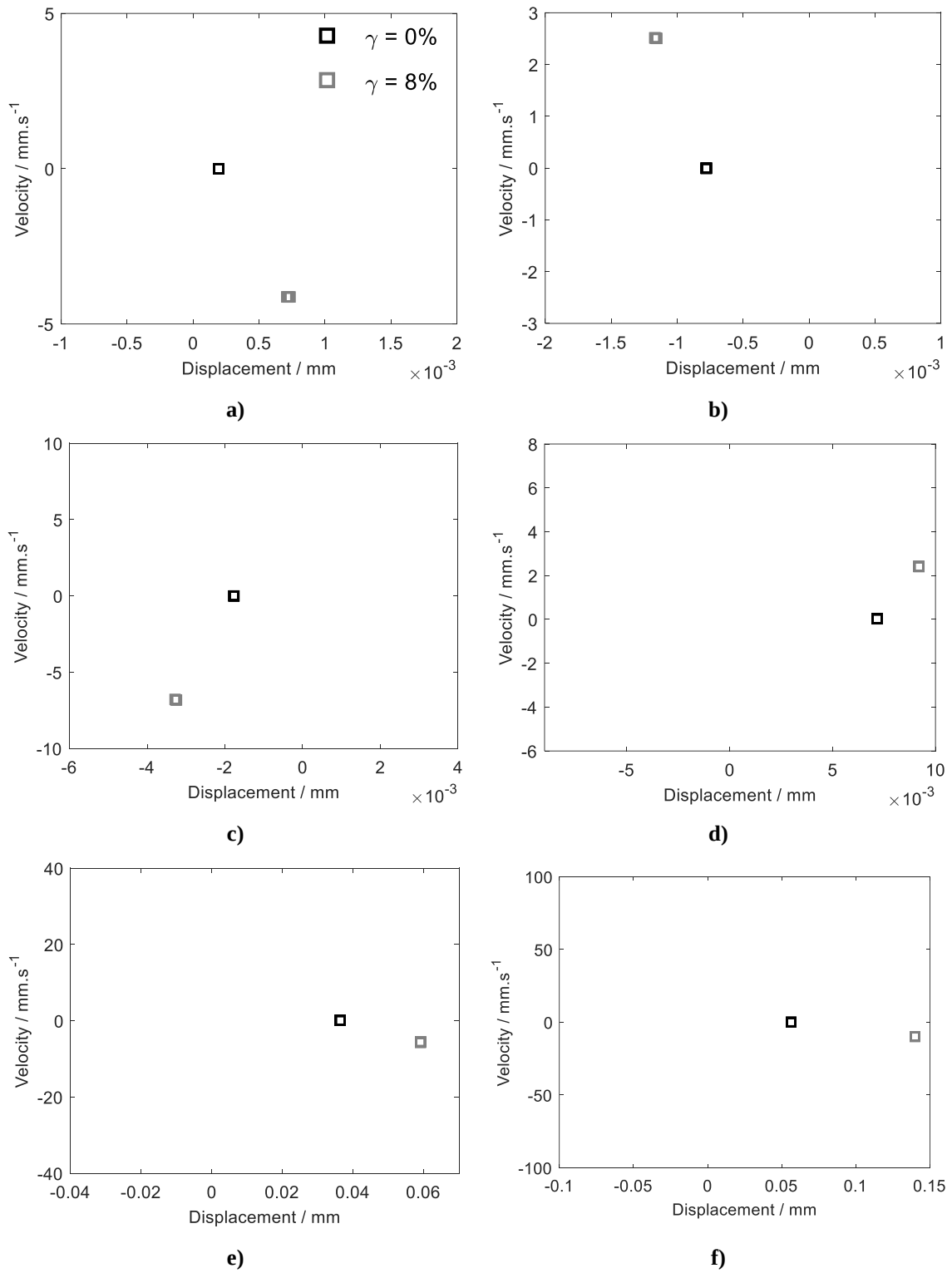


**Figure 7.18** – Poincaré sections of the dynamic analysis on the substructure with initial flaws of  $\gamma = 0\%$  and  $\gamma = 8\%$  at  $\omega = 500 \text{ rad.s}^{-1}$  in point: a) 1; b) 2; c) 3; d) 4; e) 5; f) 6.

#### 7.4.4. NUMERICAL RESULTS OF THE DYNAMIC ANALYSIS ON THE SUBSTRUCTURE – EXCITATION FREQUENCY INFLUENCE

To observe now the importance of the excitation frequency on the obtained results, the same simulation is replicated but now with  $\omega = 1000 \text{ rad.s}^{-1}$  and a duration of 1.5 s.

As the used frequency has changed, to use the same 40 discretisation points per wave, it is necessary to verify the convergence of the results again. Table 7.5 lists information about the developed simulations at  $\omega = 1000 \text{ rad.s}^{-1}$  and Figure 7.19 a) and Figure 7.19 b) present the time-history and the phase portrait, respectively, of this test. The reference simulation uses 120 points to assure accurate results.

**Table 7.5** – Time increments used, and time spent for different number of discretisation points per wave at  $\omega = 1000 \text{ rad.s}^{-1}$  in the dynamic analysis of the substructure.

| Points per Wave | $\Delta t$ [s]           | Time        |
|-----------------|--------------------------|-------------|
| 120             | $5.23599 \times 10^{-5}$ | 29 h 18 min |
| 40              | $1.57080 \times 10^{-4}$ | 10 h 26 min |

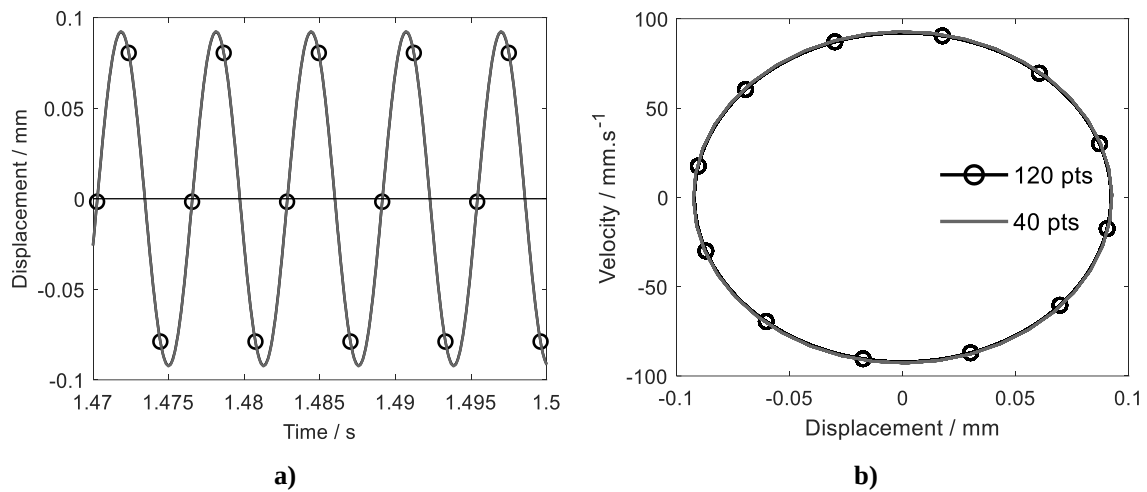


**Figure 7.19** – Time-history (a) and phase portrait (b) of the dynamic convergence study on the substructure, for a different number of discretisation points per wave, at  $\omega = 1000 \text{ rad.s}^{-1}$ .

As seen in Figure 7.19, the results with 120 and 40 discretisation points per wave are pretty much the same. Thus, the convergence with 40 points is here verified for the dynamic analysis on the healthy substructure.

Next, in Figure 7.20 the time-histories for the simulations with initial damages of  $\gamma = 0 \%$  and  $\gamma = 8 \%$  and excitation frequencies of  $\omega = 500 \text{ rad.s}^{-1}$  and  $\omega = 1000 \text{ rad.s}^{-1}$  are displayed. Right after, in Figure 7.21 is presented the time-history from the detached parts. These graphics and all others in this section are show results from the top central point of the composite panel.

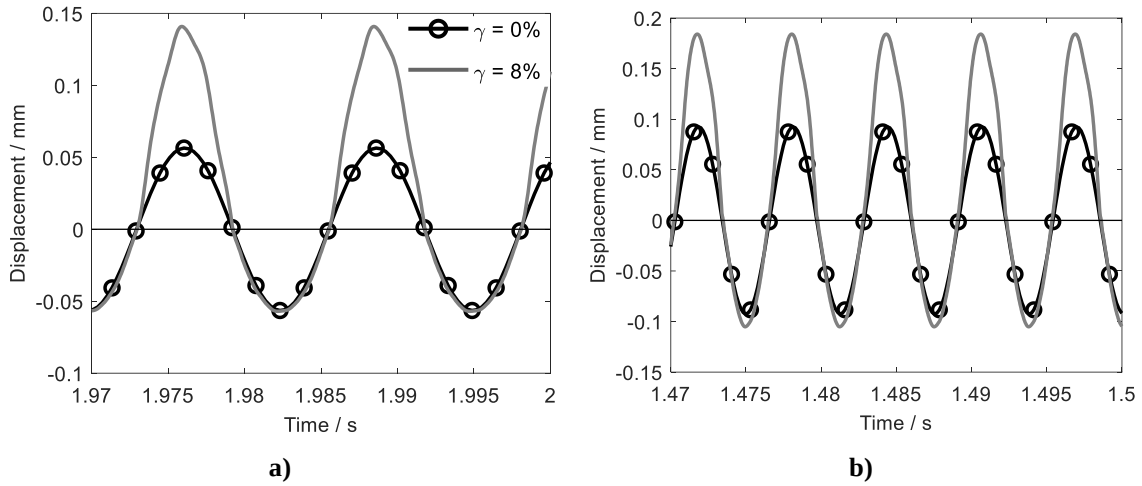


**Figure 7.20** – Time-history of the dynamic analysis on the substructure with initial flaws of  $\gamma = 0 \%$  and  $\gamma = 8 \%$  at: a)  $\omega = 500 \text{ rad.s}^{-1}$ ; b)  $\omega = 1000 \text{ rad.s}^{-1}$ .

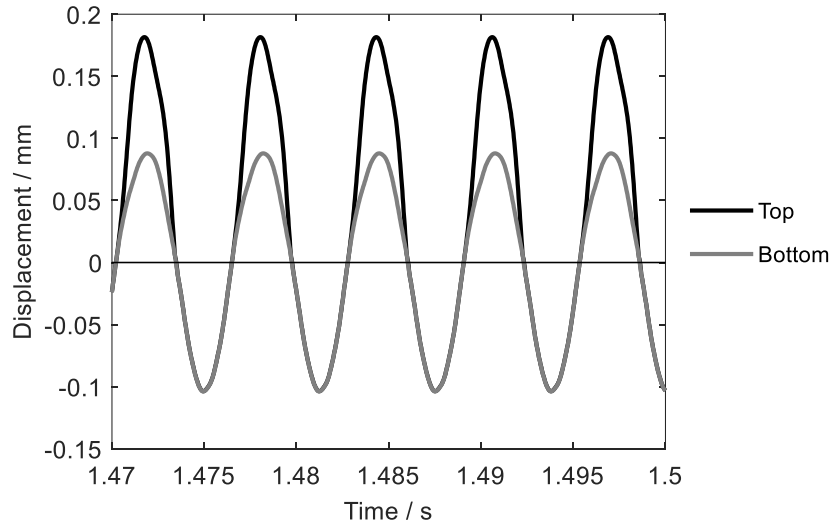


**Figure 7.21** – Time-history of the detached parts on dynamic analysis at  $\omega = 1000 \text{ rad.s}^{-1}$  for an initial flaw of  $\gamma = 8 \%$ .

First, from these figures, it is clear that in the amplitude from the sane structure's response is bigger for  $\omega = 1000 \text{ rad.s}^{-1}$  than for  $\omega = 500 \text{ rad.s}^{-1}$ . Consequently, in the simulations with an initial damage of  $\gamma = 8 \%$ , the resulting amplitude is also bigger in the simulation with  $\omega = 1000 \text{ rad.s}^{-1}$  and the variation between the overlapped results.

Just like in the analysed results from section 7.4.2, for an excitation frequency  $\omega = 1000 \text{ rad.s}^{-1}$  the presence of nonlinearities is clear, the substructure and the detached parts oscillate in the same phase and there is a contact per period of the response, with very similar characteristics.

Figure 7.22 shows the respective frequency spectrum to the time-histories from Figure 7.19.

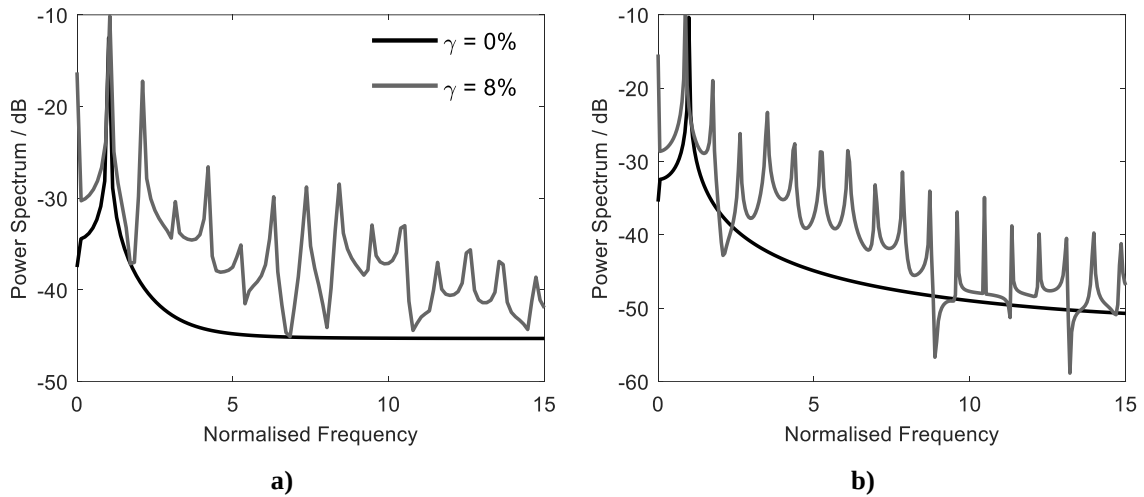


**Figure 7.22** – Frequency spectrum of the dynamic analysis on the substructure with initial flaws of  $\gamma = 0\%$  and  $\gamma = 8\%$  at: a)  $\omega = 500 \text{ rad.s}^{-1}$ ; b)  $\omega = 1000 \text{ rad.s}^{-1}$ .

Although the established contact between the detached parts is very similar in both simulations at  $\omega = 500 \text{ rad.s}^{-1}$  and  $\omega = 1000 \text{ rad.s}^{-1}$ , the generated harmonics of order superior to two are not the same. In the simulation at  $\omega = 1000 \text{ rad.s}^{-1}$ , the 3<sup>rd</sup> and 5<sup>th</sup> have a more important contribution. Also, the contribution from the harmonics decreases continuously with their order.

Lastly, Figure 7.23 represents the phase portraits of the previously mentioned simulations.

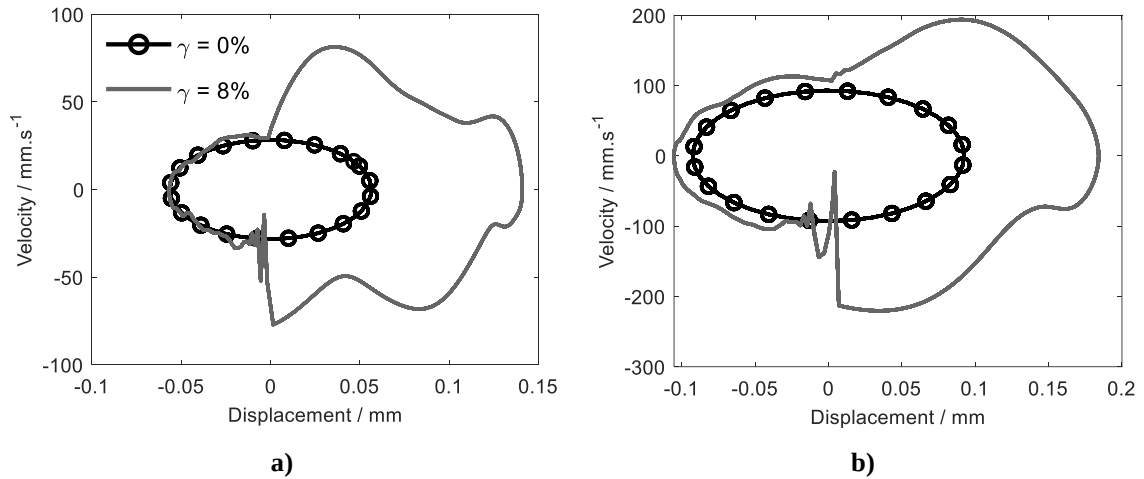


**Figure 7.23** – Phase portrait of the dynamic analysis on the substructure with initial flaws of  $\gamma = 0\%$  and  $\gamma = 8\%$  at: a)  $\omega = 500 \text{ rad.s}^{-1}$ ; b)  $\omega = 1000 \text{ rad.s}^{-1}$ .

These graphics are important to reveal the increase of the evaluated displacements and velocities in the  $\omega = 1000 \text{ rad.s}^{-1}$  simulation comparatively to the  $\omega = 500 \text{ rad.s}^{-1}$  one.

Observing the entire set of results from this section, one can conclude that the use of an excitation frequency of  $\omega = 1000 \text{ rad.s}^{-1}$  results in bigger displacement amplitudes and, hence, bigger velocities, accelerations, and variations. For measuring results in real life, all of this is very important since it becomes easier to do so.

To fully understand the influence of the excitation frequency in the results, other simulations must be performed. Also, the natural frequencies should be explored to take advantage of the resonance phenomenon.

## 7.5. FINAL REMARKS

The studied substructure is part of a larger wing representative structure which, in turn, would be part of an airplane. In these, the maintenance operations are essential to guarantee passenger safety and are frequently made using VSHM approaches.

Over this chapter, the substructure in study undergoes a thorough modal and implicit dynamic analysis. Taking them into account, the applicability of VSHM methodologies is approached now.

In a VSHM, the main interest is to identify small flaws (the smaller, the better). In the implicit dynamic analysis with a harmonic loading, the smallest flaw of  $\gamma = 0.4 \%$  is possible to detect but in fact the variation in the displacements is so small that it can go unnoticed in real life, especially if the measurements are made in terms of displacement. On the other hand, the associated velocity and acceleration variations are much more significant and if the measurements are made with an accelerometer, they are more likely to be noticed.

An identified strategy to increase the amplitude of the responses and, hence, the variation between the results is to increase the excitation frequency (and the approximation to the first natural frequency of the substructure). The use of the natural frequencies as excitation is not approached here due to convergence difficulties but seems very promising. Other strategy relies on the increase of the amplitude of the excitation load or the use of an excitation frequency that is a sub harmonic of the fundamental natural frequency. In the second scenario, if the system's response is nonlinear, the appearance of a higher-order harmonic that coincides with the fundamental natural frequency must be very noticeable. Another way to more easily detect these small flaws is using the extremity points of the composite panel, that are very sensitive to higher-order harmonics, and that can announce a flaw's presence earlier.

Additionally, the increase of the initial flaw size, makes easier to detect the presence of damage in the substructure. This is not helpful in the detection of small damages, obviously, but in the long run, if the substructure's damaged response is well known, it can be important to quantify the flaw's size.

Another very important goal of VSHM is the detection of the damage's location. The analyses to evaluate the measurement location's importance in the results, ended up with few conclusions thanks to the influence of the single point load (which could be diminished, per example, by using a distributed across the surface load or a damage parameter, like the Lyapunov exponents) and of the local stiffness variation. With these restrictions the damage location identification becomes very unlikely and the VSHM method used can only be classified as a level 1 method.

Regarding the multiple tools used throughout the previous chapters, it is important to say that, for a VSHM analysis, the time-histories and phase portraits reveal themselves very useful since they do not need a comparative response and allow to detect the presence of damage by simply exhibiting

nonlinearities in their responses. Frequency spectra and Poincaré sections also display easily the influence of nonlinearities but require the same structure's response, which can be restrictive. Also, in this specific situation, to draw the Poincaré sections it is important to know the phase portraits since the "location" of the first point influences greatly the detection or not of the existing flaw. Moreover, for all these methodologies, the identification of the damage's location is a possibility (even though it is not possible right now).

In the modal analysis, one concludes that this approach can also be used to identify the presence of damage, being that the bigger the flaw, the bigger the natural frequencies variation. However, this is the methodology in which the variations are less noticeable because it does not account for nonlinearities of the structure.

## CHAPTER 8. CONCLUSION

---

### 8.1. CONCLUSIONS

In this dissertation, a wing representative structure undergoes a thorough analysis. This structure is composed of aluminium and composite materials and assembled with hybrid joints. The multiple and varied performed analyses aim to identify the structure's fragilities, such as points or areas prone for damage onset and propagation and to conclude about the applicability of nonlinear VSHM methodologies.

First, in the static analysis, a numerical model of the wing representative structure is developed. After, the adhesive layer of Araldite® 420 A/B located between the composite panel and the long stiffeners is identified as the location most likely to fail and a new simpler substructure with limitations resembling the ones from the original structure is defined. This allows to develop more easily the progressive damage and dynamic analyses, which are more complex than the static one.

The progressive damage analysis allows to infer that the most important damage mechanisms are the yielding of the stiffeners and debond of the Araldite® 420 A/B layer. Next, one evaluates the scenario of an initial damage in the adhesive layer caused by environmental degradation or bad adhesion and sees how this promotes the damage initiation and propagation. It was observed that the plasticisation of the stiffeners occurs earlier, and the damage extent is bigger.

Lastly, in the dynamic analysis, the detection of the presence of damage is accomplished easily, even for small flaws. This detection occurs when there is non mono-harmonic responses in the time-history graphics, complex phase portraits (that are not an ellipse), frequency spectra with the influence of higher-order harmonics that are multiples of the excitation frequency and/or Poincaré sections with different results from the ones of the healthy substructure. Farther, changes in the natural frequencies and mode shapes of the system indicate the presence of damage too. However, the determination of damage's locus is not possible. Thus, in the present work, the VSHM methodologies used are classified as level 1 methods.

Additionally, the developed work in this dissertation constituted an opportunity to acquire a knowhow about the constitutive modelling of elastic, elastoplastic, and damaged materials, contact mechanics and the FEM. Moreover, it was chance to understand the functionalities and the potential of ABAQUS, a FE's commercial software, and the knowledge behind it.

## 8.2. FUTURE WORK

After all the work developed, several suggestions of future work to improve and complement the wing representative structure's analysis can be given:

- Rerun all the analysis considering now the previously neglected bolted connections and understand if these would affect the already obtained results;
- Experimentally approve all the results;
- In the last static analysis with the cohesive elements, assign them the damage propagation law of the Araldite® 420 A/B and identify the exact moment when the adhesive layer fails;
- In the progressive damage analyses, add an initial flaw located in the middle of the composite panel instead of in the adhesive connection. Even though, the second one is the critical interface, it is quite common for defects to appear within the panel due to a low velocity impact;
- In the dynamic analysis of the substructure, observe the free decay oscillations after an impulse loading and draw the frequency spectrum of the response to conclude about the true natural frequencies of the system. Compare the results with the ones of the modal analysis and understand the importance of the contact interactions between the detached parts;
- Besides the already performed analysis about the influence of the initial flaw size, the measurement location, and the excitation frequency, determine the importance of the amplitude of the excitation force and the damage location. Also, evaluate the response for higher frequencies than the ones studied and see if there are benefits in using excitations at the natural frequencies;
- Investigate the applicability of excitations at subharmonics of resonance frequencies, taking advantage of the fact that harmonics of the excitation frequency are stimulated in the response due to the damage induced nonlinearity;
- Perform the dynamic analysis where the measurement location is evaluated but now with a distributed harmonic load to see if more conclusions can be drawn. Try to apply a damage parameter such as the maximum Lyapunov exponent and see if it is possible to determine the damage location from the multiple measures (and reach a level 2 classification for the used VSHM method);
- Repeat all dynamic analysis but with the initial flaw located within the composite panel. Like previously stated, these are most likely to appear during service;
- Develop the progressive damage and the dynamic analysis in the whole structure and see if the main findings are the same.

## REFERENCES

---

- Ajibose, O. K., M. Wiercigroch, E. Pavlovskaya, and A. R. Akisanya. 2010. "Global and local dynamics of drifting oscillator for different contact force models." *International Journal of Non-Linear Mechanics* 45 (9): 850-858. <https://doi.org/10.1016/j.ijnonlinmec.2009.11.017>. <https://www.sciencedirect.com/science/article/pii/S0020746209002194>.
- Andreus, Ugo, and P. Baragatti. 2009. "Fatigue crack growth, free vibrations, and breathing crack detection of aluminium alloy and steel beams." *Journal of Strain Analysis for Engineering Design - J STRAIN ANAL ENG DESIGN* 44: 595-608. <https://doi.org/10.1243/03093247JSA527>. [https://www.researchgate.net/publication/239407930\\_Fatigue\\_crack\\_growth\\_free\\_vibrations\\_and\\_breathing\\_crack\\_detection\\_of\\_aluminium\\_alloy\\_and\\_steel\\_beams](https://www.researchgate.net/publication/239407930_Fatigue_crack_growth_free_vibrations_and_breathing_crack_detection_of_aluminium_alloy_and_steel_beams).
- CES EduPack 2020.
- Bar-Cohen, Yoseph 2000. "Emerging NDE Technologies and Challenges at the Beginning of the 3rd Millennium -- Part II." Accessed 21.04.2021. <https://www.ndt.net/article/v05n01/barcohen/barcohen.htm>.
- Bathe, K.J. 2014. *Finite Element Procedures*. 2nd ed.: Prentice Hall.
- Blackman, B. R. K., H. Hadavinia, A. J. Kinloch, and J. G. Williams. 2003. "The use of a cohesive zone model to study the fracture of fibre composites and adhesively-bonded joints." *International Journal of Fracture* 119 (1): 25-46. <https://doi.org/10.1023/A:1023998013255>. <https://link.springer.com/article/10.1023/A:1023998013255>.
- Bobrov, A. V., B. S. Sarbaev, and Yu Yu Shirshov. 2016. "Deformation and strength properties of a carbon-carbide composite with 2D reinforcement under plane stress state." *Journal of Machinery Manufacture and Reliability* 45 (2): 145-151. <https://doi.org/10.3103/S1052618816020035>.
- Bonet, Javier, and Richard D. Wood. 2008. *Nonlinear Continuum Mechanics for Finite Element Analysis*. 2nd ed. Cambridge: Cambridge University Press.
- Braga, Daniel, L. F. M. Silva, and Pedro Moreira. 2018. "Single Lap Joints Numerical Modelling and Comparison with Experimental Testing." *U.Porto Journal of Engineering* 2: 11-20. [https://doi.org/10.24840/2183-6493\\_002.001\\_0002](https://doi.org/10.24840/2183-6493_002.001_0002). [https://journalengineering.fe.up.pt/article/view/2183-6493\\_002.001\\_0002](https://journalengineering.fe.up.pt/article/view/2183-6493_002.001_0002).
- Braga, Daniel, L. Sousa, V. Infante, L. F. M. Silva, and Pedro Moreira. 2015. "Aluminium Friction Stir Weld-bonded Joints." *The Journal of Adhesion* 92: 150912073238005. <https://doi.org/10.1080/00218464.2015.1085860>. [https://www.researchgate.net/publication/283037743\\_Aluminium\\_Friction\\_Stir\\_Weld-bonded\\_Joints](https://www.researchgate.net/publication/283037743_Aluminium_Friction_Stir_Weld-bonded_Joints).
- Burlayenko, V. N., and T. Sadowski. 2010. "Influence of skin/core debonding on free vibration behavior of foam and honeycomb cored sandwich plates." *International Journal of Non-Linear Mechanics* 45 (10): 959-968. <https://doi.org/10.1016/j.ijnonlinmec.2009.07.002>. <https://www.sciencedirect.com/science/article/pii/S0020746209001309>.
- Burlayenko, V. N., and T. Sadowski. 2011a. "Dynamic behaviour of sandwich plates containing single/multiple debonding." *Computational Materials Science* 50 (4): 1263-1268. <https://doi.org/10.1016/j.commatsci.2010.08.005>. [https://www.researchgate.net/publication/248504200\\_Dynamic\\_behaviour\\_of\\_sandwich\\_plates\\_containing\\_singlemultiple\\_debonding](https://www.researchgate.net/publication/248504200_Dynamic_behaviour_of_sandwich_plates_containing_singlemultiple_debonding).
- Burlayenko, V. N., and T. Sadowski. 2011b. "Post-impact dynamic response of sandwich plates with foam and non-metallic honeycomb cores." *Proceedings of the 7th European Nonlinear Dynamics Conference ENOC*.
- Burlayenko, V. N., and T. Sadowski. 2012a. "Finite element nonlinear dynamic analysis of sandwich plates with partially detached facesheet and core." *Finite Elements in Analysis and Design* 62: 49-64. <https://doi.org/10.1016/j.finel.2012.08.003>. <https://www.sciencedirect.com/science/article/pii/S0168874X1200159X>.
- Burlayenko, V. N., and T. Sadowski. 2012b. "A numerical study of the dynamic response of sandwich plates initially damaged by low-velocity impact." *Computational Materials Science* 52 (1): 212-216. <https://doi.org/10.1016/j.commatsci.2011.01.009>. <https://www.sciencedirect.com/science/article/pii/S0927025611000280>.
- Burlayenko, V. N., and T. Sadowski. 2014. "Nonlinear dynamic analysis of harmonically excited debonded sandwich plates using finite element modelling." *Composite Structures* 108: 354-366. <https://doi.org/10.1016/j.compstruct.2013.09.042>.

- [https://www.researchgate.net/publication/262558904\\_Nonlinear\\_dynamic\\_analysis\\_of\\_harmonically\\_excited\\_debonded\\_sandwich\\_plates\\_using\\_finite\\_element\\_modelling](https://www.researchgate.net/publication/262558904_Nonlinear_dynamic_analysis_of_harmonically_excited_debonded_sandwich_plates_using_finite_element_modelling).
- Burlayenko, V. N., and T. Sadowski. 2018. "Linear and Nonlinear Dynamic Analyses of Sandwich Panels with Face Sheet-to-Core Debonding." *Shock and Vibration* 2018: 5715863. <https://doi.org/10.1155/2018/5715863>. [https://www.researchgate.net/publication/323194875\\_Linear\\_and\\_Nonlinear\\_Dynamic\\_Analyses\\_of\\_Sandwich\\_Panels\\_with\\_Face\\_Sheet-to-Core\\_Debonding](https://www.researchgate.net/publication/323194875_Linear_and_Nonlinear_Dynamic_Analyses_of_Sandwich_Panels_with_Face_Sheet-to-Core_Debonding).
- Camacho, P. S. F. 2020. "Aeroelastic Analysis of Hybrid Multiscale Composite Plates."
- Camanho, Pedro, and Carlos Dávila. 2002. "Mixed-Mode Decohesion Finite Elements for the Simulation of Delamination in Composite Materials." [https://www.researchgate.net/publication/24327361\\_Mixed-Mode\\_Decohesion\\_Finite\\_Elements\\_for\\_the\\_Simulation\\_of\\_Delamination\\_in\\_Composite\\_Materials](https://www.researchgate.net/publication/24327361_Mixed-Mode_Decohesion_Finite_Elements_for_the_Simulation_of_Delamination_in_Composite_Materials).
- Camanho, Pedro, Carlos Dávila, and Marcelo De Moura. 2003. "Numerical Simulation of Mixed-Mode Progressive Delamination in Composite Materials." *Journal of Composite Materials* 37. <https://doi.org/10.1177/0021998303034505>. [https://www.researchgate.net/publication/37650396\\_Numerical\\_Simulation\\_of\\_Mixed-Mode\\_Progressive\\_Delamination\\_in\\_Composite\\_Materials](https://www.researchgate.net/publication/37650396_Numerical_Simulation_of_Mixed-Mode_Progressive_Delamination_in_Composite_Materials).
- Carminati, M., and Sergio Ricci. 2018. "Structural Damage Detection Using Nonlinear Vibrations." *International Journal of Aerospace Engineering* 2018: 1-21. <https://doi.org/10.1155/2018/1901362>. [https://www.researchgate.net/publication/327890037\\_Structural\\_Damage\\_Detection\\_Using\\_Nonlinear\\_Vibrations](https://www.researchgate.net/publication/327890037_Structural_Damage_Detection_Using_Nonlinear_Vibrations).
- Cawley, P, and R D Adams. 1979. "The location of defects in structures from measurements of natural frequencies." *The Journal of Strain Analysis for Engineering Design* 14 (2): 49-57. <https://doi.org/10.1243/03093247v142049>.
- Charalambides, MN, AJ Kinloch, and FL Matthews. 1997. "Strength prediction of bonded joints." *Bolted/Bonded Joints in Polymeric Composites*.
- Chati, M., R. Rand, and S. Mukherjee. 1997. *MODAL ANALYSIS OF A CRACKED BEAM*. Vol. 207. Vol. 2 *Journal of Sound and Vibration*.
- Cho, Maenghyo, and Jun-Sik Kim. 2001. "Higher-Order ZigZag Theory for Laminated Composites With Multiple Delaminations." *Journal of Applied Mechanics-transactions of The Asme - J APPL MECH* 68. <https://doi.org/10.1115/1.1406959>.
- Choi, Hyung Yun, Hsi-Yung T. Wu, and Fu-Kuo Chang. 1991. "A New Approach toward Understanding Damage Mechanisms and Mechanics of Laminated Composites Due to Low-Velocity Impact: Part II—Analysis." *Journal of Composite Materials* 25 (8): 1012-1038. <https://doi.org/10.1177/002199839102500804>.
- Chondros, T. G., A. D. Dimarogonas, and J. Yao. 2001. "VIBRATION OF A BEAM WITH A BREATHING CRACK." *Journal of Sound and Vibration* 239 (1): 57-67. <https://doi.org/10.1006/jsvi.2000.3156>. <https://www.sciencedirect.com/science/article/pii/S0022460X00931560>.
- Cousigné, Olivier, David Moncayo, Daniel Coutellier, Pedro Camanho, Hakim Naceur, and Steffen Hampel. 2013. "Development of a new nonlinear numerical material model for woven composite materials accounting for permanent deformation and damage." *Composite Structures* 106: 601-614. <https://doi.org/10.1016/j.compstruct.2013.07.026>. <https://www.sciencedirect.com/science/article/pii/S0263822313003498>.
- da Silva, L. F. M., and Andreas Öchsner. 2008. *Modeling of Adhesively Bonded Joints*.
- de Moraes, A., A. Pereira, M. de Moura, F. Silva, and N. Dourado. 2015. "Bilinear approximations to the mixed-mode I–II delamination cohesive law using an inverse method." *Compos. Struct.* 122. <https://doi.org/10.1016/j.compstruct.2014.11.058>. [https://www.researchgate.net/publication/270594721\\_Bilinear\\_approximations\\_to\\_the\\_mixed-mode\\_I-II\\_delamination\\_cohesive\\_law\\_using\\_an\\_inverse\\_method](https://www.researchgate.net/publication/270594721_Bilinear_approximations_to_the_mixed-mode_I-II_delamination_cohesive_law_using_an_inverse_method).
- de Moura, M. F. S. F., and J. A. G. Chousal. 2006. "Cohesive and continuum damage models applied to fracture characterization of bonded joints." *International Journal of Mechanical Sciences* 48 (5): 493-503. <https://doi.org/10.1016/j.ijmecsci.2005.12.008>. <https://www.sciencedirect.com/science/article/pii/S0020740305002997>.
- de Moura, M. F. S. F., R. Daniaud, and A. G. Magalhães. 2006. "Simulation of mechanical behaviour of composite bonded joints containing strip defects." *Int J Adhes Adhes* 26 (6): 464-473. <https://doi.org/10.1016/j.ijadhadh.2005.06.010>. <https://www.sciencedirect.com/science/article/pii/S0143749605001119>.
- de Moura, M. F. S. F., J. P. M. Gonçalves, A. T. Marques, and P. M. S. T. de Castro. 2000. "Prediction of compressive strength of carbon–epoxy laminates containing delamination by using a mixed-mode damage model." *Composite Structures* 50 (2): 151-157. [https://doi.org/10.1016/S0263-8223\(00\)00091-X](https://doi.org/10.1016/S0263-8223(00)00091-X). [https://www.researchgate.net/publication/222528489\\_Prediction\\_of\\_compressive\\_strength\\_of\\_carbon-epoxy\\_laminates\\_containing\\_delamination\\_by\\_using\\_a\\_mixed-mode\\_damage\\_model](https://www.researchgate.net/publication/222528489_Prediction_of_compressive_strength_of_carbon-epoxy_laminates_containing_delamination_by_using_a_mixed-mode_damage_model).
- de Moura, Marcelo, António Magalhães, and A. B. Moraes. 2018. *Materiais Compósitos - Materiais, Fabrico e Comportamento Mecânico*.
- Dourado, N., M. F. S. F. de Moura, A. B. de Moraes, and A. B. Pereira. 2012. "Bilinear approximations to the mode II delamination cohesive law using an inverse method." *Mechanics of Materials* 49: 42-50.

- <https://doi.org/10.1016/j.mechmat.2012.02.004>.  
<https://www.sciencedirect.com/science/article/pii/S0167663612000361>.
- Elmalich, Dvir, and Oded Rabinovitch. 2015. "On the effect of inter-laminar contact on the dynamics of locally delaminated FRP strengthened walls." *International Journal of Non-Linear Mechanics* 77: 141-157.  
<https://doi.org/10.1016/j.ijnonlinmec.2015.07.003>.  
<https://www.sciencedirect.com/science/article/pii/S0020746215001298>.
- Epureanu, Bogdan, Michael Paidoussis, and Liaosha Tang. 2004. "Exploiting Chaotic Dynamics for Detecting Parametric Variations in Aeroelastic Systems." *Aiaa Journal - AIAA J* 42: 728-735.  
<https://doi.org/10.2514/1.9556>.
- European COST. 2019. "Cost Action CA18203 - Optimizing Design for Inspection (ODIN)." Accessed 02/12/2020. <http://odin-cost.com/>.
- Fu, Shao-Yun, Xiao Hu, and Chee-Yoon Yue. 1998. "A New Model for the Transverse Modulus of Unidirectional Fiber Composites." *Journal of Materials Science* 33 (20): 4953-4960.  
<https://doi.org/10.1023/A:1004442520797>. <https://link.springer.com/article/10.1023/A:1004442520797>.
- Gonçalves, J., Marcelo De Moura, Antônio Magalhães, and P. Castro. 2003. "Application of interface element to the three-dimensional progressive failure analysis of adhesive joints." *Fatigue & Fracture of Engineering Materials & Structures* 26: 479-486. <https://doi.org/10.1046/j.1460-2695.2003.00653.x>.  
[https://www.researchgate.net/publication/227888173\\_Application\\_of\\_interface\\_element\\_to\\_the\\_three-dimensional\\_progressive\\_failure\\_analysis\\_of\\_adhesive\\_joints](https://www.researchgate.net/publication/227888173_Application_of_interface_element_to_the_three-dimensional_progressive_failure_analysis_of_adhesive_joints).
- Gonçalves, J. P. M., Marcelo De Moura, P. M. S. T. Castro, and Antônio Marques. 2000. "Interface element including point-to-surface constraints for three-dimensional problems with damage propagation." *Engineering Computations* 17: 28-47. <https://doi.org/10.1108/02644400010308053>.  
[https://www.researchgate.net/publication/243461394\\_Interface\\_element\\_including\\_point-to-surface\\_constraints\\_for\\_three-dimensional\\_problems\\_with\\_damage\\_propagation](https://www.researchgate.net/publication/243461394_Interface_element_including_point-to-surface_constraints_for_three-dimensional_problems_with_damage_propagation).
- Gordan, Meisam, Hashim Abdul Razak, Zubaidah Ismail, and Khaled Ghaedi. 2017. "Recent Developments in Damage Identification of Structures Using Data Mining." *Latin American Journal of Solids and Structures* 14: 2373-2401. <https://doi.org/10.1590/1679-78254378>.  
[https://www.researchgate.net/publication/322040724\\_Recent\\_Developments\\_in\\_Damage\\_Identification\\_of\\_Structures\\_Using\\_Data\\_Mining](https://www.researchgate.net/publication/322040724_Recent_Developments_in_Damage_Identification_of_Structures_Using_Data_Mining).
- Guillaumat, L., and Z. Hamdoun. 2006. "Reliability model of drilled composite materials." *Composite Structures* 74 (4): 467-474. <https://doi.org/10.1016/j.compstruct.2005.05.002>.
- Harris, J. A., and R. A. Adams. 1984. "Strength prediction of bonded single lap joints by non-linear finite element methods." *International Journal of Adhesion and Adhesives* 4 (2): 65-78.  
[https://doi.org/10.1016/0143-7496\(84\)90103-9](https://doi.org/10.1016/0143-7496(84)90103-9).
- Hart-Smith, L. J. 1998. "Predictions of a generalized maximum-shear-stress failure criterion for certain fibrous composite laminates." *Composites Science and Technology* 58 (7): 1179-1208.  
[https://doi.org/10.1016/S0266-3538\(97\)00193-0](https://doi.org/10.1016/S0266-3538(97)00193-0).  
<https://www.sciencedirect.com/science/article/pii/S0266353897001930>.
- Hilber, Hans M., Thomas J. R. Hughes, and Robert L. Taylor. 1977. "Improved numerical dissipation for time integration algorithms in structural dynamics." *Earthquake Engineering & Structural Dynamics* 5 (3): 283-292. <https://doi.org/10.1002/eqe.4290050306>.
- Hilditch, Tim, Dale Atwell, Mark Easton, and Matthew Barnett. 2009. "Performance of wrought aluminium and magnesium alloy tubes in three-point bending." *Materials & Design* 30 (7): 2316-2322.  
<https://doi.org/10.1016/j.matdes.2008.11.020>.  
<https://www.sciencedirect.com/science/article/pii/S0261306908005517>.
- Holzapfel, Gerhard A. 2002. *Nonlinear Solid Mechanics: A Continuum Approach for Engineering Science*. Vol. 37. Vol. 4Meccanica.
- Hu, Hsuan-Teh, Wen-Pin Lin, and Fang-Tai Tu. 2015. "Failure analysis of fiber-reinforced composite laminates subjected to biaxial loads." *Composites Part B: Engineering* 83: 153-165.  
<https://doi.org/10.1016/j.compositesb.2015.08.045>.  
<https://www.sciencedirect.com/science/article/pii/S1359836815004758>.
- Hu, N., H. Fukunaga, M. Kameyama, Y. Aramaki, and F. K. Chang. 2002. "Vibration analysis of delaminated composite beams and plates using a higher-order finite element." *International Journal of Mechanical Sciences* 44 (7): 1479-1503. [https://doi.org/10.1016/S0020-7403\(02\)00026-7](https://doi.org/10.1016/S0020-7403(02)00026-7).  
<https://www.sciencedirect.com/science/article/pii/S0020740302000267>.
- Huntsman. 2012. SAFETY DATA SHEET: ARALEDITE® 420 A/B.
- Isbilir, Ozden, and Elahieh Ghassemieh. 2013. "Numerical investigation of the effects of drill geometry on drilling induced delamination of carbon fiber reinforced composites." *Composite Structures* 105: 126-133.  
<https://doi.org/10.1016/j.compstruct.2013.04.026>.  
<https://www.sciencedirect.com/science/article/pii/S0263822313001785>.
- Johnson, K. L. 1985. *Contact Mechanics*. Cambridge: Cambridge University Press.
- Kashtalyan, Maria. 2010. *Delamination Behaviour of Composites* Vol. 114. *The Aeronautical Journal*.

- Kim, Heung Soo, Aditi Chattopadhyay, and Anindya Ghoshal. 2003. "Characterization of delamination effect on composite laminates using a new generalized layerwise approach." *Computers & Structures* 81 (15): 1555-1566. [https://doi.org/10.1016/S0045-7949\(03\)00150-0](https://doi.org/10.1016/S0045-7949(03)00150-0).  
<https://www.sciencedirect.com/science/article/pii/S0045794903001500>.
- Kim, Hyeung-Yun, and Woonbong Hwang. 2002. "Effect of debonding on natural frequencies and frequency response functions of honeycomb sandwich beams." *Composite Structures* 55 (1): 51-62. [https://doi.org/10.1016/S0263-8223\(01\)00136-2](https://doi.org/10.1016/S0263-8223(01)00136-2).  
[https://www.researchgate.net/publication/222220470\\_Effect\\_of\\_debonding\\_on\\_natural\\_frequencies\\_and\\_frequency\\_response\\_functions\\_of\\_honeycomb\\_sandwich\\_beams](https://www.researchgate.net/publication/222220470_Effect_of_debonding_on_natural_frequencies_and_frequency_response_functions_of_honeycomb_sandwich_beams).
- Kim, N.H. 2015. *Introduction to Nonlinear Finite Element Analysis*. 1st ed.: Springer US.
- Kim, Young, Kyeongsik Woo, Hyunjun Cho, In-Gul Kim, and Jun Hyun Kim. 2015. "High-Velocity Impact Damage Behavior of Carbon/Epoxy Composite Laminates." *IJASS* 16: 190-205. <https://doi.org/10.5139/IJASS.2015.16.2.190>.  
<https://www.sciencedirect.com/science/article/pii/S2214785318307764>.
- Kohar, Christopher, Amir Zhumagulov, Abhijit Brahme, Michael Worswick, Raja Mishra, and Kaan Inal. 2016. "Development of high crush efficient, extrudable aluminum front rails for vehicle lightweighting." *International Journal of Impact Engineering* 95. <https://doi.org/10.1016/j.ijimpeng.2016.04.004>.  
[https://www.researchgate.net/publication/301759950\\_Development\\_of\\_high\\_crush\\_efficient\\_extrudable\\_aluminum\\_front\\_rails\\_for\\_vehicle\\_lightweighting](https://www.researchgate.net/publication/301759950_Development_of_high_crush_efficient_extrudable_aluminum_front_rails_for_vehicle_lightweighting).
- Korelc, Joze, and Peter Wriggers. 2016. *Automation of Finite Element Methods*.
- Kwon, Y. 2002. "Dynamic Numerical Modeling and Simulation of Interfacial Cracks in Sandwich Structures for Damage Detection." *Journal of Sandwich Structures & Materials - J SANDW STRUCT MATER* 4: 175-199. <https://doi.org/10.1177/1099636202004002706>.  
<https://journals.sagepub.com/doi/10.1177/1099636202004002706>.
- Lammerant, L., and I. Verpoest. 1996. "Modelling of the interaction between matrix cracks and delaminations during impact of composite plates." *Composites Science and Technology* 56 (10): 1171-1178. [https://doi.org/10.1016/S0266-3538\(96\)00071-1](https://doi.org/10.1016/S0266-3538(96)00071-1).  
<https://www.sciencedirect.com/science/article/pii/S0266353896000711>.
- Laursen, Tod. 2003. *Computational Contact and Impact Mechanics*. Vol. 38. *Meccanica*.
- Lee, Su Jeong, and Dai Gil Lee. 1992. "Development of a Failure Model for the Adhesively Bonded Tubular Single Lap Joint." *The Journal of Adhesion* 40 (1): 1-14. <https://doi.org/10.1080/00218469208030467>.
- Lestari, Wahyu, and Pizhong Qiao. 2005. "Damage detection of fiber-reinforced polymer honeycomb sandwich beams." *Composite Structures* 67 (3): 365-373. <https://doi.org/10.1016/j.compstruct.2004.01.023>.  
[https://www.researchgate.net/publication/228358282\\_Damage\\_detection\\_of\\_fiber-reinforced\\_polymer\\_honeycomb\\_sandwich\\_beams](https://www.researchgate.net/publication/228358282_Damage_detection_of_fiber-reinforced_polymer_honeycomb_sandwich_beams).
- Li, Qunhong, Yuming Chen, Yanhui Wei, and Qishao Lu. 2011. "The analysis of the spectrum of Lyapunov exponents in a two-degree-of-freedom vibro-impact system." *International Journal of Non-Linear Mechanics* 46 (1): 197-203. <https://doi.org/10.1016/j.ijnonlinmec.2010.08.009>.  
[https://www.researchgate.net/publication/223461519\\_The\\_analysis\\_of\\_the\\_spectrum\\_of\\_Lyapunov\\_exponents\\_in\\_a\\_two-degree-of-freedom\\_vibro-impact\\_system](https://www.researchgate.net/publication/223461519_The_analysis_of_the_spectrum_of_Lyapunov_exponents_in_a_two-degree-of-freedom_vibro-impact_system).
- Luo, H., and S. Hanagud. 2000. "Dynamics of delaminated beams." *International Journal of Solids and Structures* 37 (10): 1501-1519. [https://doi.org/10.1016/S0020-7683\(98\)00325-4](https://doi.org/10.1016/S0020-7683(98)00325-4).  
<https://www.sciencedirect.com/science/article/pii/S0020768398003254>.
- Maia, N. M. M., J. M. M. Silva, E. A. M. Almas, and R. P. C. Sampaio. 2003. "DAMAGE DETECTION IN STRUCTURES: FROM MODE SHAPE TO FREQUENCY RESPONSE FUNCTION METHODS." *Mechanical Systems and Signal Processing* 17 (3): 489-498. <https://doi.org/10.1006/mssp.2002.1506>.  
<https://www.sciencedirect.com/science/article/pii/S0888327002915062>.
- Maimí, P., P. P. Camanho, J. A. Mayugo, and C. G. Dávila. 2007. "A continuum damage model for composite laminates: Part II – Computational implementation and validation." *Mechanics of Materials* 39 (10): 909-919. <https://doi.org/10.1016/j.mechmat.2007.03.006>.  
<https://www.sciencedirect.com/science/article/pii/S0167663607000555>.
- Manoach, Emil, Sylwester Samborski, Andrzej Mitura, and Jerzy Warminski. 2012. "Vibration based damage detection in composite beams under temperature variations using Poincaré maps." *International Journal of Mechanical Sciences* 62 (1): 120-132. <https://doi.org/10.1016/j.ijmecsci.2012.06.006>.  
<https://www.sciencedirect.com/science/article/pii/S0020740312001312>.
- Manoach, Emil, and Irina Trendafilova. 2008. "Large amplitude vibrations and damage detection of rectangular plates." *Journal of Sound and Vibration* 315 (3): 591-606. <https://doi.org/10.1016/j.jsv.2008.02.016>.  
<https://www.sciencedirect.com/science/article/pii/S0022460X08001521>.
- Manoach, Emil, Jerzy Warminski, Lukasz Kloda, and Andrzej Teter. 2017. "Numerical and experimental studies on vibration based methods for detection of damage in composite beams." *Composite Structures* 170: 26-39. <https://doi.org/10.1016/j.compstruct.2017.03.005>.
- Marjanović, Miroslav, Günther Meschke, and Djordje Vuksanović. 2016. "A finite element model for propagating delamination in laminated composite plates based on the Virtual Crack Closure method."

- Composite Structures* 150: 8-19. <https://doi.org/10.1016/j.compstruct.2016.04.044>.  
<https://www.sciencedirect.com/science/article/pii/S0263822316303877>.
- Marjanović, Miroslav, Djordje Vuksanović, and Günther Meschke. 2015. "Geometrically nonlinear transient analysis of delaminated composite and sandwich plates using a layerwise displacement model with contact conditions." *Composite Structures* 122: 67-81. <https://doi.org/10.1016/j.compstruct.2014.11.028>.  
<https://www.sciencedirect.com/science/article/pii/S0263822314005972>.
- Martiny, Ph, F. Lani, A. J. Kinloch, and T. Pardoen. 2013. "A maximum stress at a distance criterion for the prediction of crack propagation in adhesively-bonded joints." *Engineering Fracture Mechanics* 97: 105-135. <https://doi.org/10.1016/j.engfracmech.2012.10.025>.  
<https://www.sciencedirect.com/science/article/pii/S001379441200433X>.
- Mirmohammad, S. H., M. Safarabadi, M. Karimpour, M. R. M. Aliha, and F. Berto. 2018. "Study of Composite Fiber Reinforcement of Cracked Thin-Walled Pressure Vessels Utilizing Multi-Scaling Technique Based on Extended Finite Element Method." *Strength of Materials* 50 (6): 925-936. <https://doi.org/10.1007/s11223-019-00041-4>.  
<https://link.springer.com/article/10.1007/s11223-019-00041-4>.
- Mitsubishi Chemical. 2020. "Pyrofil." Accessed 05-02-2021. [https://www.m-chemical.co.jp/en/products/departments/mcc/cfcm/product/CFtow\\_Jan2020en.pdf](https://www.m-chemical.co.jp/en/products/departments/mcc/cfcm/product/CFtow_Jan2020en.pdf).
- Mkaddem, Ali, Ibrahim Demirci, and Mohamed El Mansori. 2008. "A micro-macro combined approach using FEM for modelling of machining of FRP composites: Cutting forces analysis." *Composites Science and Technology* 68 (15): 3123-3127. <https://doi.org/10.1016/j.compscitech.2008.07.009>.  
[https://www.researchgate.net/publication/248407459\\_A\\_micro-macro\\_combined\\_approach\\_using\\_FEM\\_for\\_modelling\\_of\\_machining\\_of\\_FRP\\_composites\\_Cutting\\_forces\\_analysis](https://www.researchgate.net/publication/248407459_A_micro-macro_combined_approach_using_FEM_for_modelling_of_machining_of_FRP_composites_Cutting_forces_analysis).
- Moreira, P. M. G. P., P. F. P. de Matos, P. P. Camanho, Stefan D. Pastrama, and P. M. S. T. de Castro. 2007. "Stress intensity factor and load transfer analysis of a cracked riveted lap joint." *Materials & Design* 28 (4): 1263-1270. <https://doi.org/10.1016/j.matdes.2005.12.014>.  
<https://www.sciencedirect.com/science/article/pii/S0261306906000094>.
- Moreira, Raul, Marcelo De Moura, Filipe Silva, Francis Mariana Gonzalez Ramirez, and J. Rodrigues. 2020. "Mixed-mode I+II fracture characterisation of composite bonded joints." *Journal of Adhesion Science and Technology* 34: 1-14. <https://doi.org/10.1080/01694243.2019.1708645>.
- Motamedi, D., and S. Mohammadi. 2010. "Dynamic analysis of fixed cracks in composites by the extended finite element method." *Engineering Fracture Mechanics* 77 (17): 3373-3393. <https://doi.org/10.1016/j.engfracmech.2010.08.011>.  
[https://www.researchgate.net/publication/223548381\\_Dynamic\\_analysis\\_of\\_fixed\\_cracks\\_in\\_composites\\_by\\_the\\_extended\\_finite\\_element\\_method](https://www.researchgate.net/publication/223548381_Dynamic_analysis_of_fixed_cracks_in_composites_by_the_extended_finite_element_method).
- Naik, Niranjana, and Chandra Yerramalli. 1998. "Damage in Laminated Composites Due to Low Velocity Impact." *Journal of Reinforced Plastics and Composites* 17: 1232-1263. <https://doi.org/10.1177/073168449801701401>.  
[https://www.researchgate.net/publication/258156745\\_Damage\\_in\\_Laminated\\_Composites\\_Due\\_to\\_Low\\_Velocity\\_Impact](https://www.researchgate.net/publication/258156745_Damage_in_Laminated_Composites_Due_to_Low_Velocity_Impact).
- Nayfeh, A., and B. Balachandran. 1994. *Applied Nonlinear Dynamics : Analytical, Computational, and Experimental Methods*. New York: John Wiley and Sons.
- Nichols, J. M., C. J. Nichols, M. D. Todd, M. Seaver, S. T. Trickey, and L. N. Virgin. 2004. "Use of data-driven phase space models in assessing the strength of a bolted connection in a composite beam." *Smart Materials and Structures* 13 (2): 241-250. <https://doi.org/10.1088/0964-1726/13/2/001>.
- Nuismer, R.J., and J.D. Labor. 1978. "Applications of the Average Stress Failure Criterion: Part I — Tension." *Journal of Composite Materials* 12 (3): 238-249. <https://doi.org/10.1177/002199837801200302>.
- Nuismer, R.J., and J.D. Labor. 1979. "Applications of the Average Stress Failure Criterion: Part II — Compression." *Journal of Composite Materials* 13 (1): 49-60. <https://doi.org/10.1177/002199837901300104>.
- Oliver, Javier, and Carlos Agelet de Saracibar. 2017. *Continuum Mechanics for Engineers. Theory and Problems. Second Edition*. 3rd ed.
- Petyt, Maurice. 2010. *Introduction to Finite Element Vibration Analysis*. 2 ed. Cambridge: Cambridge University Press.
- Pinninti, Ravinder Reddy, and Ramamurthy.G. 1997. "Stress Intensity Factor using Finite Element Analysis in Rectangular Orthotropic Composite Annular Disk." *Defense Science Journal* 47: pp.55-63. <https://doi.org/10.14429/dsj.47.3977>.  
[https://www.researchgate.net/publication/274385586\\_Stress\\_Intensity\\_Factor\\_using\\_Finite\\_Element\\_Analysis\\_in\\_Rectangular\\_Orthotropic\\_Composite\\_Annular\\_Disk](https://www.researchgate.net/publication/274385586_Stress_Intensity_Factor_using_Finite_Element_Analysis_in_Rectangular_Orthotropic_Composite_Annular_Disk).
- Popp, Alexander. 2012. "Mortar Methods for Computational Contact Mechanics and General Interface Problems." Ph. D., Technische Universität München.
- Raju, Benjamin, S. R. Hiremath, and D. Roy Mahapatra. 2018. "A review of micromechanics based models for effective elastic properties of reinforced polymer matrix composites." *Composite Structures* 204: 607-619. <https://doi.org/10.1016/j.compstruct.2018.07.125>.  
<https://www.sciencedirect.com/science/article/pii/S0263822318315411>.

- Reddy, J. N. 2013. *An Introduction to Continuum Mechanics*. 2 ed. Cambridge: Cambridge University Press.
- Ribeiro, F. M. F., R. D. S. G. Campilho, R. J. C. Carbas, and L. F. M. da Silva. 2016. "Strength and damage growth in composite bonded joints with defects." *Composites Part B: Engineering* 100: 91-100. <https://doi.org/10.1016/j.compositesb.2016.06.060>.  
<https://www.sciencedirect.com/science/article/pii/S1359836816310332>.
- Shin, K. C., and J. J. Lee. 2003. "Bond Parameters Affecting Failure of Co-Cured Single and Double Lap Joints Subjected to Static and Dynamic Tensile Loads." In *European Structural Integrity Society*, edited by B. R. K. Blackman, A. Pavan and J. G. Williams, 373-384. Elsevier.
- SK Chemicals. 2013. "K51 Epoxy Prepreg." Accessed 05-02-2021. [http://www.triplehcomposites.co.uk/userfiles/files/SKR%20K51%20ver1\\_0\(1\).pdf](http://www.triplehcomposites.co.uk/userfiles/files/SKR%20K51%20ver1_0(1).pdf).
- Smith, M. 2009a. "6.3.5 Natural frequency extraction." In *ABAQUS Analysis User's Manual, Version 6.9*. Dassault Systèmes Simulia Corp.
- Smith, M. 2009b. "17.2.3 Plane stress orthotropic failure measures." In *ABAQUS Analysis User's Manual, Version 6.9*. Dassault Systèmes Simulia Corp.
- Smith, M. 2009c. "19.3.1 Damage and failure for fiber-reinforced composites: overview." In *ABAQUS Analysis User's Manual, Version 6.9*. Dassault Systèmes Simulia Corp.
- Smith, M. 2009d. "19.3.3 Damage evolution and element removal for fiber-reinforced composites." In *ABAQUS Analysis User's Manual, Version 6.9*. Dassault Systèmes Simulia Corp.
- Smith, M. 2009e. "22.1.1 Solid (continuum) elements." In *ABAQUS Analysis User's Manual, Version 6.9*. Dassault Systèmes Simulia Corp.
- Smith, M. 2009f. "29.2.1 Defining contact pairs in ABAQUS/Standard." In *ABAQUS Analysis User's Manual, Version 6.9*. Dassault Systèmes Simulia Corp.
- Smith, R. 2009g. "Composite defects and their detection." *Materials Science and Engineering* 3. <http://www.eolss.net/sample-chapters/c05/e6-36-04-03.pdf>.
- Song, Jeong-Hoon, and Ted Belytschko. 2009. "Cracking node method for dynamic fracture with finite elements." *International Journal for Numerical Methods in Engineering* 77 (3): 360-385. <https://doi.org/10.1002/nme.2415>.  
[https://www.researchgate.net/publication/227544165\\_Cracking\\_node\\_method\\_for\\_dynamic\\_fracture\\_with\\_finite\\_elements](https://www.researchgate.net/publication/227544165_Cracking_node_method_for_dynamic_fracture_with_finite_elements).
- The British Standards Institution. 2006. *Cold formed welded structural hollow sections of non-alloy and fine grain steels. Part 1: Technical delivery conditions*: BSI Standards.
- The British Standards Institution. 2019. *Hot rolled products of structural steels. Part 2: Technical delivery conditions for non-alloy structural steels*: BSI Standards.
- Thompson, JMT, and HB Stewart. 2001. *Nonlinear Dynamics and Chaos*. Wiley: New York.
- Trendafilova, Iirina. 2006. "Vibration-based damage detection in structures using time series analysis." *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* 220 (3): 261-272. <https://doi.org/10.1243/09544062c12904>.  
[https://www.researchgate.net/publication/229004682\\_VIBRATION-BASED\\_DAMAGE\\_DETECTION\\_USING\\_TIME\\_SERIES\\_ANALYSIS](https://www.researchgate.net/publication/229004682_VIBRATION-BASED_DAMAGE_DETECTION_USING_TIME_SERIES_ANALYSIS).
- Trendafilova, Iirina, and Emil Manoach. 2012. "Nonlinear dynamics/time series methods for fault detection and identification in structural and robotic systems." 111-138.
- Tsyfansky, S. L., and V. I. Beresnevich. 2000. "NON-LINEAR VIBRATION METHOD FOR DETECTION OF FATIGUE CRACKS IN AIRCRAFT WINGS." *Journal of Sound and Vibration* 236 (1): 49-60. <https://doi.org/10.1006/jsvi.2000.2981>.  
[https://www.researchgate.net/publication/243365667\\_Non-linear\\_vibration\\_method\\_for\\_detection\\_of\\_fatigue\\_cracks\\_in\\_aircraft\\_wings](https://www.researchgate.net/publication/243365667_Non-linear_vibration_method_for_detection_of_fatigue_cracks_in_aircraft_wings).
- Ugen, Alexandre, Lierni Zubillaga, Albert Turon, and Nicolas Carrere. 2014. *Comparison of cohesive zone models used to predict delamination initiated from free-edges: Validation against experimental results*.
- Vasiliev, Valery V., and Evgeny V. Morozov. 2007. "Chapter 6 - FAILURE CRITERIA AND STRENGTH OF LAMINATES." In *Advanced Mechanics of Composite Materials (Second Edition)*, edited by Valery V. Vasiliev and Evgeny V. Morozov, 321-357. Oxford: Elsevier Science Ltd.
- Voigt, W. 1889. "Ueber die Beziehung zwischen den beiden Elasticitätsconstanten isotroper Körper." *Annalen der Physik* 274 (12): 573-587. <https://doi.org/10.1002/andp.18892741206>.
- Wang, J. T. S., Y. Y. Liu, and J. A. Gibby. 1982. "Vibrations of split beams." *Journal of Sound and Vibration* 84 (4): 491-502. [https://doi.org/10.1016/S0022-460X\(82\)80030-8](https://doi.org/10.1016/S0022-460X(82)80030-8).
- Wang, Jian, and L. Tong. 2002. "A study of the vibration of delaminated beams using a nonlinear anti-interpenetration constraint model." *Composite Structures* 57: 483-488. [https://doi.org/10.1016/S0263-8223\(02\)00117-4](https://doi.org/10.1016/S0263-8223(02)00117-4).  
[https://www.researchgate.net/publication/222633666\\_A\\_study\\_of\\_the\\_vibration\\_of\\_delaminated\\_beams\\_using\\_a\\_nonlinear\\_anti-interpenetration\\_constraint\\_model](https://www.researchgate.net/publication/222633666_A_study_of_the_vibration_of_delaminated_beams_using_a_nonlinear_anti-interpenetration_constraint_model).
- Wilson, Dale W., and R. Byron Pipes. 1981. "Analysis of the Shearout Failure Mode in Composite Bolted Joints." In *Composite Structures*, edited by I. H. Marshall, 34-49. Dordrecht: Springer Netherlands.

- Wolfe, Howard, Cynthia Shroyer, Dansen Brown, and Larry Simmons. 1995. "An Experimental Investigation of Nonlinear Behaviour of Beam and Plates Excited to High Levels of Dynamic Response." Southampton Univ. (United Kingdom).
- Worden, K., Gr Tomlinson, and Kazuyuki Yagasaki. 2002. "Nonlinearity in Structural Dynamics: Detection, Identification and Modeling." *Applied Mechanics Reviews - APPL MECH REV* 55. <https://doi.org/10.1115/1.1451161>.
- Worden, Keith, Charles R. Farrar, Jonathan Haywood, and Michael Todd. 2008. "A review of nonlinear dynamics applications to structural health monitoring." *Structural Control and Health Monitoring* 15 (4): 540-567. <https://doi.org/10.1002/stc.215>. [https://www.researchgate.net/publication/227763174\\_A\\_review\\_of\\_nonlinear\\_dynamic\\_applications\\_to\\_structural\\_health\\_monitoring](https://www.researchgate.net/publication/227763174_A_review_of_nonlinear_dynamic_applications_to_structural_health_monitoring).
- Wriggers, Peter. 2006. *Computational Contact Mechanics*.
- Xie, De, and Anthony M. Waas. 2006. "Discrete cohesive zone model for mixed-mode fracture using finite element analysis." *Engineering Fracture Mechanics* 73 (13): 1783-1796. <https://doi.org/10.1016/j.engfracmech.2006.03.006>. <https://www.sciencedirect.com/science/article/pii/S0013794406001007>.
- Yang, Q. D., M. D. Thouless, and S. M. Ward. 1999. "Numerical simulations of adhesively-bonded beams failing with extensive plastic deformation." *Journal of the Mechanics and Physics of Solids* 47 (6): 1337-1353. [https://doi.org/10.1016/S0022-5096\(98\)00101-X](https://doi.org/10.1016/S0022-5096(98)00101-X). <https://ui.adsabs.harvard.edu/abs/1999JMPSo..47.1337Y/abstract>.
- Yang, Shu, and Chang Qi. 2013. "Multiobjective optimization for empty and foam-filled square columns under oblique impact loading." *International Journal of Impact Engineering* 54: 177-191. <https://doi.org/10.1016/j.ijimpeng.2012.11.009>. <https://www.sciencedirect.com/science/article/pii/S0734743X12002321>.
- Żak, A., M. Krawczuk, and W. Ostachowicz. 2001. "Vibration of a Laminated Composite Plate with Closing Delamination." *Journal of Intelligent Material Systems and Structures* 12 (8): 545-551. <https://doi.org/10.1177/10453890122145320>. <https://journals.sagepub.com/doi/abs/10.1177/10453890122145320>.
- Zienkiewicz, O. C., and R. L. Taylor. 2000a. *The Finite Element Method: Solid Mechanics*. Vol. 2nd. edited by Butterworth-Heinemann.
- Zienkiewicz, O. C., and R. L. Taylor. 2000b. *The Finite Element Method: The Basis*. Vol. 1st. edited by Butterworth-Heinemann.



# APPENDIX A CALCULUS OF THE LAMINATE'S ENGINEERING CONSTANTS WITH THE CLASSICAL LAMINATED PLATE THEORY

---

The composite in question has a laminated structure. To understand it as a whole, first, it is necessary to characterise every elemental ply. For that purpose, de Moura et al. (2018) was followed.

Each of these plies is perceived as a unidirectional composite with a high enough FC that allows it to be treated as a homogeneous material (its properties are independent of the scope's location). However, properties are still a function of the direction of measurement and the USN150B is analysed as transversely isotropic.

It is important to highlight that the principal material directions of a lamina are defined such that axis 1 is along the fibres, axis 2 is perpendicular to axis 1 and in the lamina's plane and axis 3 is along the thickness – Figure A.1.

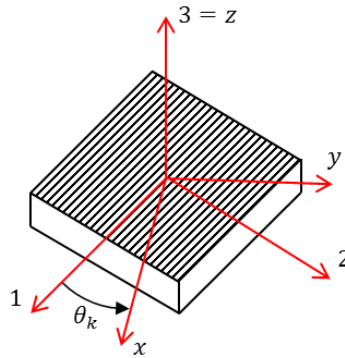


**Figure A.1** – Coordinate systems used.

In this coordinate system, the stiffness matrix  $\mathbb{K}$  that establishes the linear stress-strain relation is the same as Equation (3.34).

Each layer  $k$  has its own fibre's orientation  $\theta_k$ , so to analyse the entire laminate, it is necessary to rotate and obtain this relation at a general  $xyz$  coordinate system (also in Figure A.1). The superscript  $k$ , used from now on, refers to variables that are unique for each layer, because they are dependent of the fibre's orientation. For this, a stress transformation matrix from the  $xyz$  coordinate system to the  $123$  ( $\mathbf{T}^k$ ) is established as

$$\mathbf{T}^k = \begin{bmatrix} m^2 & n^2 & 0 & 0 & 0 & 2mn \\ n^2 & m^2 & 0 & 0 & 0 & -2mn \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & m & -n & 0 \\ 0 & 0 & 0 & n & m & 0 \\ -mn & mn & 0 & 0 & 0 & m^2 - n^2 \end{bmatrix}, \quad (\text{A.1})$$

where  $m = \cos \theta_k$  and  $n = \sin \theta_k$ . Using this last matrix, it is possible to calculate the stiffness matrix that establishes the stress-strain relation in the xyz coordinate system, for each layer as

$$\bar{\mathbb{K}}^k = \mathbf{T}^{k-1} \mathbb{K} \mathbf{T}^{kT}. \quad (\text{A.2})$$

Finally, it is possible to determine the stiffness matrices  $\mathbb{A}$ ,  $\mathbb{B}$  and  $\mathbb{D}$  (using the simplification of Equation (3.33)) according to

$$(A_{ij}, B_{ij}, D_{ij}) = \sum_{k=1}^{n_l} \int_{z_{k-1}}^{z_k} (1, z, z^2) \bar{K}_{ij}^k, \quad (\text{A.3})$$

where  $n_l$  is the total number of layers and  $z_{k-1}$  and  $z_k$  are the coordinates of the bottom and upper surfaces of the layer  $k$ .

The compliance matrix  $\mathbb{a}$  is

$$\mathbb{a} = \mathbb{A}^{-1}, \quad (\text{A.4})$$

and the engineering constants of the laminate, considering the total laminate thickness  $h$ , are

$$E_x = \frac{1}{ha_{11}}, \quad (\text{A.5})$$

$$E_y = \frac{1}{ha_{22}}, \quad (\text{A.6})$$

$$G_{xy} = \frac{1}{ha_{66}}, \quad (\text{A.7})$$

$$\nu_{xy} = -\frac{a_{12}}{a_{11}}, \quad (\text{A.8})$$

$$\nu_{yx} = -\frac{a_{12}}{a_{22}}. \quad (\text{A.9})$$

For USN150B, with stacking  $(-45, 0, 45, 0, 90, 0, -45, 0, 45)_s$ , the calculated Young's modulus along the  $x$  direction is  $E_x = 70.97$  GPa.

