COMILLAS
UNIVERSIDAD PONTIFICIA
ICAI

MÁSTER EN INGENIERÍA INDUSTRIAL

TRABAJO FIN DE MÁSTER

Strength analysis of scarf-repaired composite laminates using FE modelling

Autor: Jorge Ortega Torremocha

Director: Xiao Chen

Co-Directores: Xingyuan Miao

Seyedsina Samarehmousavi

Aura Venessa de Guzman Paguagan

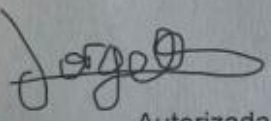
Madrid

Declaro, bajo mi responsabilidad, que el Proyecto presentado con el título

.....Strength analysis of scarf-repaired composite laminates....

en la ETS de Ingeniería - ICAI de la Universidad Pontificia Comillas en el curso académico ...2022 - 2023... es de mi autoría, original e inédito y no ha sido presentado con anterioridad a otros efectos. El Proyecto no es plagio de otro, ni total ni parcialmente y la información que ha sido tomada de otros documentos está debidamente referenciada.

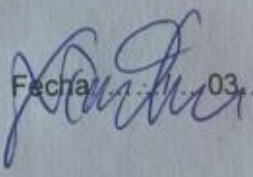
Fdo.: Jorge Ortega Torremocha Fecha: ...09.../ ...03.../ ...2023...



Autorizada la entrega del proyecto

EL DIRECTOR DEL PROYECTO

Fdo.: Xiao Chen Fecha: ...09.../ ...03.../ ...2023...



COMILLAS
UNIVERSIDAD PONTIFICIA
ICAI

MÁSTER EN INGENIERÍA INDUSTRIAL

TRABAJO FIN DE MÁSTER

Strength analysis of scarf-repaired composite laminates using FE modelling

Autor: Jorge Ortega Torremocha

Director: Xiao Chen

Co-Directores: Xingyuan Miao

Seyedsina Samarehmousavi

Aura Venessa de Guzman Paguagan

Madrid

STRENGTH ANALYSIS OF SCARF-REPAIRED COMPOSITE LAMINATES USING FE MODELLING

Autor: Ortega Torremocha, Jorge.
Director: Chen, Xiao.
Entidad Colaboradora: DTU- Denmark Technical University

RESUMEN DEL PROYECTO

Como trabajo principal en este proyecto se desarrolla una herramienta automatizada de generación de modelos de elementos finitos. El nombre de la herramienta creada es Multi Parameter Step Scarf o MPSS. El programa simula una lámina de material compuesto reparada de un aerogenerador. Las láminas han sido reparadas mediante el método de step scarf repair.

Palabras clave: Step Scarf Repair, aerogenerador, elementos finitos.

1. Introducción

Las turbinas eólicas tienen una vida estimada de 20 – 40 años según la antigüedad de la turbina. Durante su vida útil las palas de las turbinas sufren distintos fenómenos que afectan a su integridad mecánica como el granizo o la erosión del perfil aerodinámico frontal, y como consecuencia, han de ser reparadas. Una forma de repararlas es el step scarf repair o reparación escalonada del orificio. En este proyecto se estudia cómo esta reparación afecta a las palas, y, en específico, cuál es su tensión de rotura tras la reparación.

2. Definición del proyecto

Este proyecto continuo el algoritmo creado por Aura Venessa Paguagan. Con el algoritmo se puede crear de manera automática un modelo de elementos finitos de una lámina reparada mediante el step scarf repair según las características que se hayan puesto en el algoritmo. Este proyecto continuo la creación de este algoritmo mejorándolo y añadiendo distintas opciones para que el usuario pueda crear más variedad de reparaciones y pueda obtener más variedad de resultados. Las partes principales de este proyecto han sido la creación de una interfaz para el programa, la implementación del fallo de la lámina por tensión y por momento y la validación del programa mediante datos experimentales existentes.

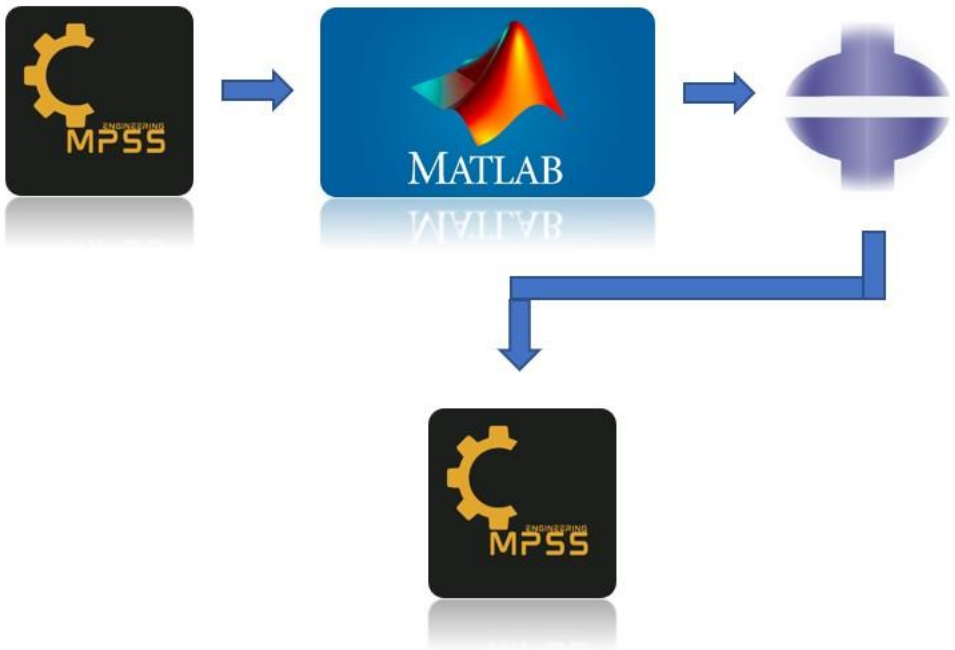
El algoritmo crea los modelos de manera automatizada reduciendo drásticamente el tiempo necesario para optimizar una reparación. El método de reparación step scarf es utilizado comúnmente en la industria de la energía eólica para reparar las palas de los aerogeneradores. Además del desarrollo de la herramienta, se ha creado una interfaz gráfica de usuario (GUI) del algoritmo. El fin del programa creado es ayudar a diseñar este tipo de reparaciones y optimizarlas sin tener que rediseñarlas manualmente ahorrando así tiempo y dinero.

Mas concretamente, en este proyecto se crea un programa que calcula la resistencia de una lámina hecha de distintas capas de fibras reparada mediante el método de step scarf repair. Para el estudio se realiza una modelización por elementos finitos. En el artículo se investiga cómo los parámetros geométricos y las propiedades de los materiales que afectan a la calidad de la reparación. Los resultados de las simulaciones se han validado con los datos experimentales existentes.

3. Descripción del modelo/sistema/herramienta

El programa creado denominado Multi Parameter Step Scarf (MPSS) realiza 4 procesos principales:

1. Input: el usuario crea el modelo de elementos finitos con las especificaciones que desee usando la interfaz creada (GUI).
2. Creación del modelo: el algoritmo crea el modelo de elementos finitos de la lámina. Este proceso se realiza usando MATLAB.
3. Run: el modelo de elementos finitos se manda al programa de elementos finitos llamado ABAQUS. El programa ABAQUS simula la lámina con las fuerzas deseadas.
4. Output: el MPSS lee los resultados del ABAQUS. Una vez que se han leído los resultados de ABAQUS, se calculan los resultados pedidos en la interfaz con las funciones correspondientes. Estos resultados son mostrados en la interfaz en forma de gráficas y datos numéricos.

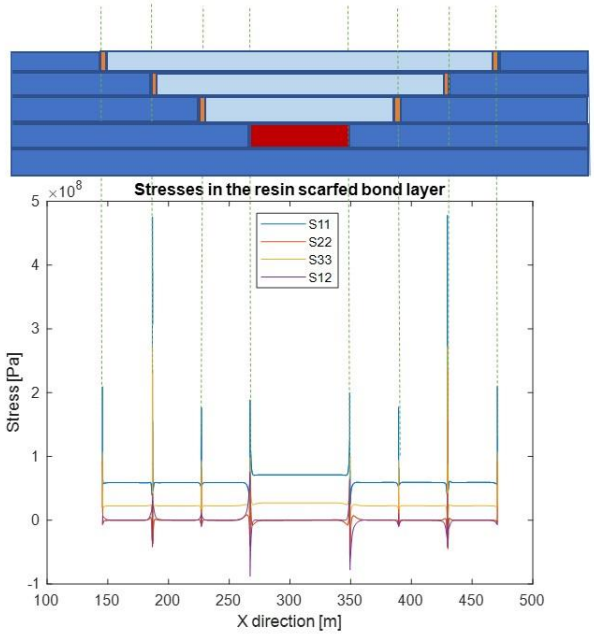


Resumen 1: Procedimiento del programa

4. Resultados

Los resultados se comparan con datos experimentales para dar validez al programa y al modelo. Los resultados son comparados a dos niveles:

1. Para comprobar que la rigidez ha sido calculada de manera correcta, se usan las tensiones obtenidas en ABAQUS. Los resultados de ABAQUS se comparan con los resultados con artículos científicos de manera cualitativa.



Resumen 2: Resultados sobre tensiones tras la reparación.

Las conclusiones a las que llega el artículo científico [1] son las mismas que las obtenidas con el programa por lo que el programa es validado.

2. Para comprobar que la carga ultima es calculada de manera correcta, se calculan los esfuerzos a partir del ABAQUS. Estos esfuerzos son comparados cualitativamente con los obtenidos en artículos científicos para validar esta parte del modelo.

5. Conclusiones

El programa creado funciona de manera correcta. Los datos experimentales de otros científicos lo corroboran. Además, el programa es útil para la industria ya que reduce el tiempo y el esfuerzo necesario para optimizar este tipo de reparaciones. El programa ha sido presentado en la WindEurope Annual event 2023.

6. Referencias

[1] Qing Han Bin Liu Wei Xie. “The tension failure mechanisms of composite stepped bonding repairs and joints for the aircraft primary load-bearing structures”. In: Journal of Adhesion Science and Technology (2019), pp. 1–17.

TÍTULO DEL TFG (EN INGLÉS)

Author: Ortega Torremocha, Jorge.
Supervisor Chen, Xiao.
Collaborating Entity: DTU- Denmark Technical University

ABSTRACT

The main focus of this project is to create an automated finite element model generation tool. The name of the tool created is Multi Parameter Step Scarf or MPSS. The program simulates a repaired composite laminate. The laminates have been repaired using the step scarf repair method.

Keywords: Step Scarf Repair, wind turbine, finite element.

1. Introduction

Wind turbines have an estimated life of 20 - 40 years depending on the age of the turbine. During their useful life, turbine blades suffer different phenomena that affect their mechanical integrity, such as hail or erosion of the front airfoil, and as a consequence, they have to be repaired. One way to repair them is the step scarf repair. This project studies how this repair affects the blades and, specifically, what is their failure stress after the repair.

2. Project description

This project continues the algorithm created by Aura Venessa Paguagan. The algorithm is used to automatically create a finite element model of a repaired laminate using the step scarf repair according to the characteristics that have been put into the algorithm. This project continues the creation of this algorithm by improving it and adding different options so that the user can create more variety of repairs and can obtain more variety of results. The main parts of this project have been the creation of an interface for the program, the implementation of the stress and moment failure of the laminate, and the validation of the program using existing experimental data.

The algorithm creates the models in an automated manner drastically reducing the time required to optimize a repair. The step scarf repair method is commonly used in the wind energy industry to repair wind turbine blades. In addition to the development of the tool, a graphical user interface (GUI) of the algorithm has been created. The purpose of the program created is to help design this type of repairs and optimize them without having to redesign them manually, thus saving time and money.

More specifically, in this project a program is created that calculates the resistance of a laminate made of different layers of fibers repaired by the step scarf repair method. For the study a finite element modeling is performed. In the paper it is investigated how geometrical parameters and material properties affect the quality of the repair. The results of the simulations have been validated with existing experimental data.

3. Description of the model & processes

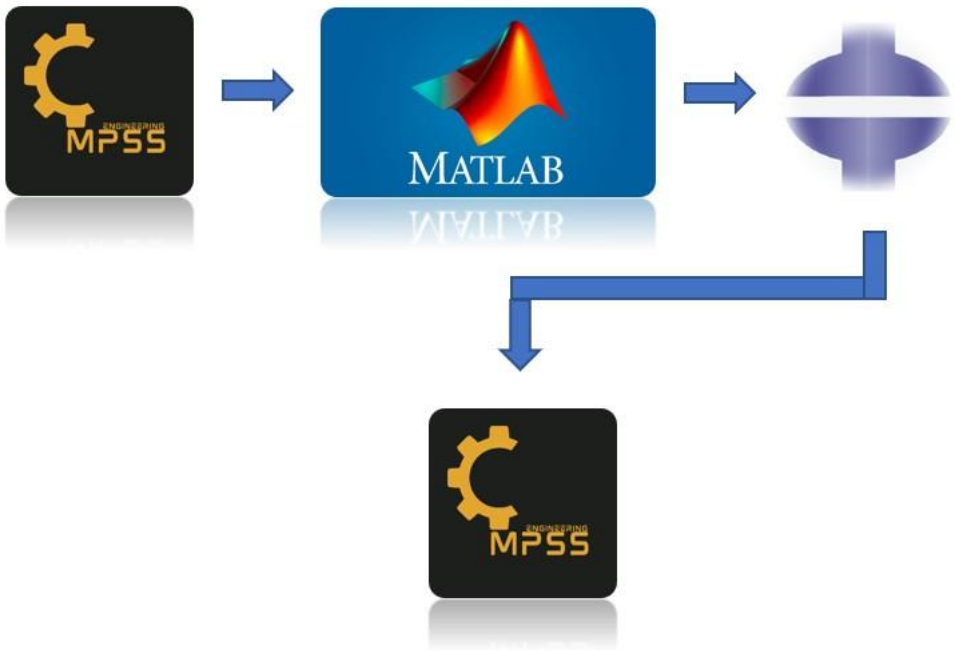
The created program called Multi Parameter Step Scarf (MPSS) performs 4 main processes:

1. Input: the user creates the finite element model with the desired specifications using the created interface (GUI).

2. Model creation: the algorithm creates the finite element model of the sheet. This process is performed using MATLAB.

3. Run: the finite element model is sent to the finite element program called ABAQUS. The ABAQUS program simulates the sheet with the desired forces.

4. Output: the MPSS reads the ABAQUS results. Once the ABAQUS results have been read, the requested results are calculated on the interface with the corresponding functions. These results are displayed on the interface in the form of graphs and numerical data.

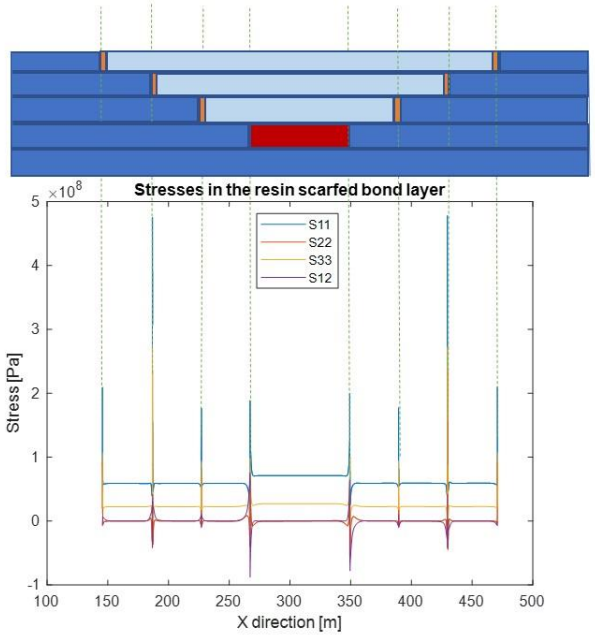


Summary 3: Program process

4. Results

The results are compared with experimental data to validate the program and the model. The results are compared at two levels:

1. To check that the stiffness has been calculated correctly, the stresses obtained in ABAQUS are used. The ABAQUS results are compared with the results of scientific papers in a qualitative way.



Summary 4: stresses after the repair

The conclusions reached by the scientific article [1] are the same as those obtained with the program, so the program is validated.

2. To check that the ultimate load is calculated correctly, stresses are calculated from the ABAQUS. These stresses are qualitatively compared with those obtained from scientific papers to validate this part of the model.

5. Conclusions

The program created works correctly. The experimental data from other scientists validates the program. In addition, the program is useful for the wind energy industry as it reduces the time and effort needed to optimize the repairs. The program was presented at the WindEurope Annual event 2023.

6. References

[1] Qing Han Bin Liu Wei Xie. “The tension failure mechanisms of composite stepped bonding repairs and joints for the aircraft primary load-bearing structures”. In: Journal of Adhesion Science and Technology (2019), pp. 1–17.

Abstract

The step scarf repair scheme is a laminate structural repair method that depends on numerous variables. This study continues the study of Aura Venessa De Guzman Paguagan [13]. In her MSc thesis, Aura discusses the effect that the different geometric values and mechanical material properties have on the final stiffness of the laminate after the repair.

From now on, the Multi-Parameter Stepped Scarf, or MPSS, is an algorithm created by Aura that allows the user to create a 2D repaired laminate input file for ABAQUS. The user can change parameters such as ply layup, ply thickness, resin thickness, mismatch, overlapping layer length, overlamina addition, load, and length of the laminate. The algorithm has been improved by adding a cohesive resin option and creating a graphical user interface, or GUI, from now on. This GUI has been compiled to create a program that anyone can download and use on their computer.

This MSc thesis aims to create a GUI of the MPSS to help future scientists study the scarf repair method. The engineers will be able to study the stiffness and the strength of the laminate after the repair. To show the usefulness of the developed tool, the GUI has been used to evaluate the strength of the repaired laminates. The results have been checked by comparing them with the research of other scientists.

The variable that has been studied is beta, which is the ratio between the thickness of the plies and the overlapping layer length of the repaired plies. The results have been discussed with the help of a load vs. displacement curve of the repaired laminate. The different failure stages of the repaired laminate have been studied in detail.

In future work, improvements to the program are discussed related to adding new materials to the GUI, implementing an optimization tool, implementing a 3D element option, and improving the range of the mismatch variable. section 6

Keywords: MPSS, GUI, step scarf repair, beta, mismatch, overlapping layer length, overlamina addition, overlamina overlapping layer length.

COMILLAS UNIVERSIDAD PONTIFICIA ICAI ICADE CIHS

Acknowledgements

I want to thank my family, specially my father and mother, for supporting me in studying and living in Denmark for one and a half years.

I would also like to show my appreciation to my tutors during the master's thesis. First, I would like to thank Aura Venessa De Guzman Paguagan (Wind Power Lab) for all our meetings on Tuesdays and for all the time you have spent with me explaining your previous work. I would also like to render my thanks to Sina Samareh (DTU Wind) for all the time spent showing me how to use ABAQUS and to model the laminate correctly. Finally, I would also like to show my appreciation to Xiao Chen (DTU Wind) and Xing-Yuan Miao (DTU Wind) for all the Friday meetings we have had during the semester to check the evolution of the thesis and for guiding me through the process.

Contents

List of Figures **6**

List of Tables **7**

1 Introduction **8**
1.1 Future estimations of the wind energy sector 9
1.2 Why are composite materials used? 10
1.3 Recycling wind turbines 10
1.4 Types of repairs in composite materials 11
1.5 Project focus 11

2 Algorithm concept **13**

3 GUI development **15**
3.1 General input 16
3.2 Geometry 17
3.3 Material properties 18
3.4 Cohesive material properties & General Static Analysis & Step control . . . 19
3.5 Run/Output 20
3.6 Output: Stress and Strain 22
3.7 Guide 1 and Guide 2 23

4 FE model **24**
4.1 SI unit system 24
4.2 Model functions 24
4.3 Material properties 26
4.4 Loading and boundary conditions 27
4.5 Element types 28
4.6 Numerical set-up and computer information 30

5 Results **31**
5.1 Validation of the homogeneous resin element model (CPE4) 31
5.1.1 Laminate geometry 31
5.1.2 Results validation 32
5.2 Validation of the cohesive resin model (COH2D4) 34
5.2.1 Laminate geometry 34
5.2.2 β parameter theoretical explanation 34
5.2.3 Failure process 35
5.2.4 Mesh convergence 44
5.2.5 Results validation 45

6 Conclusions and Future work **46**
6.1 Conclusions 46

6.2 Future work 46

7 GUI documentation **47**
7.1 General Input 47
7.1.1 Input file name 47
7.1.2 Output file name 47
7.1.3 Ply by ply match 47
7.1.4 Width defect 47
7.1.5 Overlamina addition on top of the repair 48
7.1.6 Curve panel 48
7.1.7 Resin element 48
7.1.8 Loading 48
7.1.9 Gaussian parameters guess 48
7.1.10 Number of CPU cores used for the simulation 49
7.2 Geometry 49
7.2.1 Layup selection process 49
7.2.2 Overlapping layer length 49
7.2.3 Define material thickness 50
7.2.4 Geometry of the 2D model 50
7.2.5 Element number 51
7.2.6 Discretasing density 51
7.3 Material properties 51
7.4 Cohesive material properties, General Static Analysis and Step control . . . 51
7.4.1 General Static Analysis 51
7.4.2 Cohesive elements behavior 52
7.4.3 Damage initiation criterion: QUADS & MAXS 52
7.4.4 Damage evolution: Fracture energy. 53
7.4.5 Linear elastic traction-separation behavior 53
7.4.6 Damage 54
7.4.7 Step control equilibrium: Time incrementation 54
7.5 Run/Output 55
7.5.1 MENUa and output 55
7.5.2 Plot 57
7.6 Output: Stress and Strain 58
7.7 GUI interface 59
7.8 Appdesigner 59
7.9 GUI sections 59
7.10 Loading and boundary conditions 64

8 Stiffness functions **65**
8.1 Equivalent stiffness function 65
8.2 Bending stiffness function 66

9 Sustainable Development Goals **67**

9.0.1 7^{th} Affordable and Clean Energy 67
9.0.2 13^{th} Climate Action 68

10 GUI code **68**

11 Appendix D: Run and Read functions **69**
11.1 Run function 69
11.2 Read function 70
11.3 Resources 72

List of Figures

1 LCOE 2009-2022 . . . 8
2 Lagging growth of the wind energy sector [7] . . . 9
3 Stringers [12]. . . . 10
4 Step scarf repair scheme [13] . . . 11
5 Concept . . . 14
6 General input . . . 16
7 Geometry . . . 17
8 Material properties . . . 18
9 Cohesive material properties, General Static Analysis and Step control . . . 19
10 Run and Output . . . 20
11 Output Stress and Strain . . . 22
12 Guide1 . . . 23
13 Guide2 . . . 23
14 BC for tension . . . 27
15 BC for bending . . . 27
16 Element sets in a pristine model [13] . . . 28
17 Element sets with overlamina [13] . . . 28
18 Element sets [13] . . . 29
19 Element set with mismatch [4] . . . 29
20 Step scarf path [13] . . . 31
21 Geometry for the homogeneous validation . . . 31
22 BC for the homogeneous element model validation . . . 32
23 Stress in vertical resin elements from the GUI odb results . . . 32
24 Stresses along the resin path . . . 33
25 β geometry . . . 34
26 Shear between two bonded plies by resin [6] . . . 35
27 Failure mechanism . . . 35
28 Damage initiation . . . 36
29 Traction separation law . . . 37
30 Mix mode response in cohesive elements . . . 37
31 Crack appearance in the first step . . . 38
32 Fracture modes . . . 39
33 Peel stress . . . 39
34 Shear stress . . . 40
35 Crack appearance in the second step . . . 41
36 Crack meeting point . . . 42
37 Non linearity . . . 42
38 Total failure of the laminate . . . 43
39 Video of the delamination . . . 43
40 Mesh size convergence . . . 44
41 β effect to the strength of the laminate . . . 45
42 Mismatch representation [13] . . . 47

43 Curvature formula . . . 48
44 Overlapping layer length [13] . . . 50
45 Different thicknesses [13] . . . 50
46 Mix mode response in cohesive elements [17] . . . 52
47 Fracture energy [17] . . . 53
48 GUI MENU . . . 55
49 Output . . . 56
50 MPSS Plot . . . 57
51 General input . . . 59
52 Geometry . . . 60
53 Material properties . . . 60
54 Cohesive material properties . . . 61
55 Run and Output . . . 61
56 Output: Stress and Strain . . . 62
57 Guide 1 . . . 63
58 Guide 2 . . . 63
59 Guide 3 . . . 64
60 BC for tension . . . 65
61 BC for bending . . . 65
62 Sine therorem . . . 66
63 Sustainable development goals [18] . . . 67
64 MPSS . . . 68

List of Tables

1 SI unit system . . . 24
2 Material properties . . . 26
3 Cohesive material properties . . . 26
4 Young's modulus variables . . . 65
5 Bending stiffness variables . . . 66

1 Introduction

> "I will do anything that is covered by the law to reduce Berkshire's tax rate. For example, on wind energy, we get a tax credit if we build a lot of wind farms. That's the only reason to build them. They don't make sense without the tax credit."
>
> Warren Buffett 2014.

This quote represents how much the wind energy sector has developed over the years. Nowadays, wind energy has an Levelized Cost of Energy (LCOE) of 38$ in the USA [19], making it one of the most competitive energy resources.

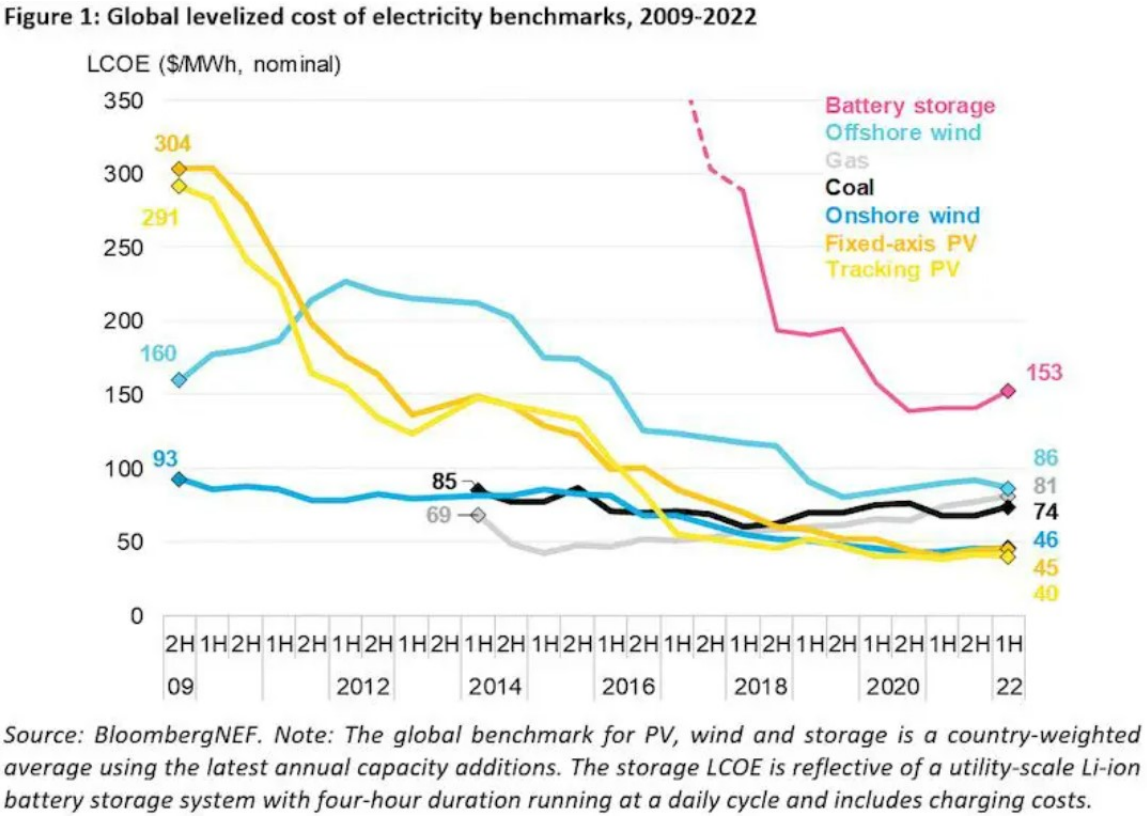


Figure 1: LCOE 2009-2022

The positive evolution (Figure 1) of the wind energy industry is undeniable. This study contributes to a certain extent, even if it is a small step in developing the wind energy industry. It will help to design and make better repairs in the future. By having higher quality repairs, the wind energy industry becomes more competitive compared to other energy sources, such as oil & gas. As a result, the improvement boosts the industry, and even if it is a far-fetched consequence, it contributes to a cleaner and less polluted atmosphere.

1.1 Future estimations of the wind energy sector

The estimated life cycle of a wind turbine blade is 20 years [3], after which the wind turbine blades have to be decommissioned. There are numerous wind farms around the globe in this day and age. More precisely, 824GW of electric wind capacity is installed, which amounts to over 300,000 wind turbines worldwide. This is the current state, and the capacity installed is only expected to grow in the future.[14]

The overall growth of the wind energy sector in 2021 was 94 GW [7] despite the setback of the pandemic. However, we are not reaching the goals set; this fact is criticized in the Intergovernmental Panel on Climate Change's (IPCC) AR6 report. At the current installation rate, in 2030, we will have less than to thirds of the required wind energy capacity installed to be on time with the net zero pathway plan.[7]

In addition, the war between Russia and Ukraine in 2022 has been a wake-up call to Europe because of its gas and oil dependency. As a result, the old coal mines in Germany have been reopened to ensure the energy supply. This is only an example of what has been happening all around Europe this year. The coal mines have been reopened; therefore, more CO2 has been introduced into the atmosphere. [10] This temporary complication adds to the bigger problem, which is not reaching the climate goals. It is estimated that the amount of wind energy installed should be higher, as stated in Figure 2.

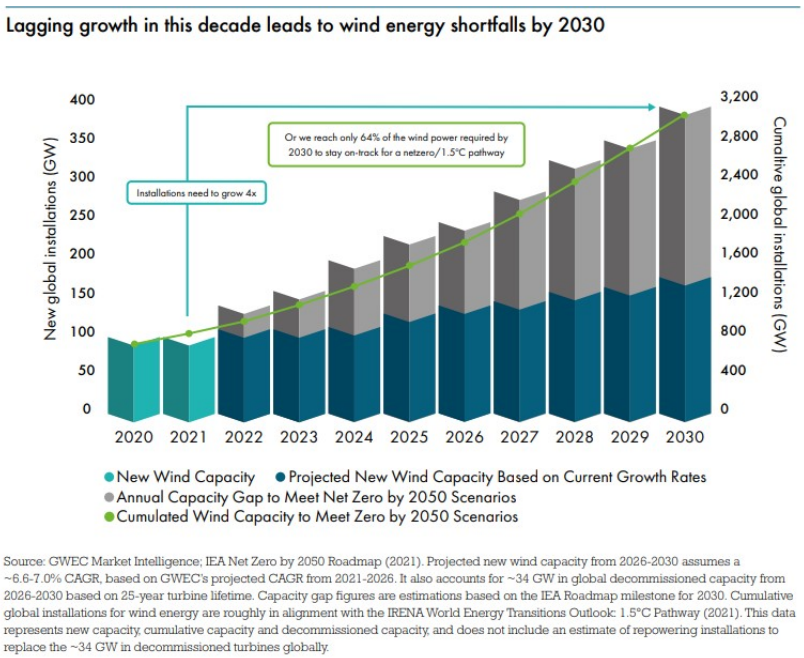
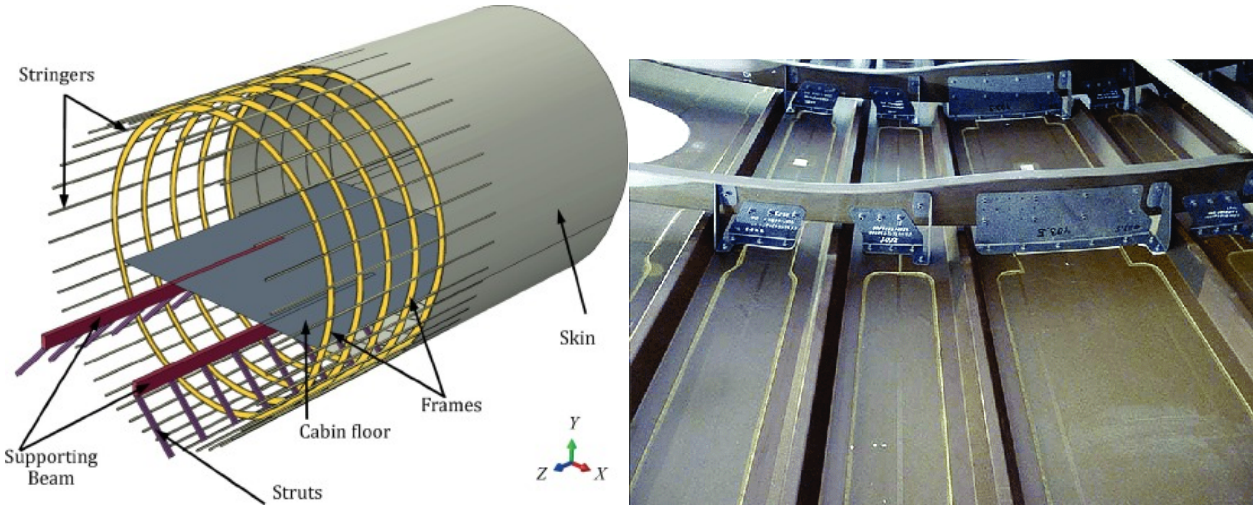


Figure 2: Lagging growth of the wind energy sector [7]

1.2 Why are composite materials used?

Composite materials are used in various industries. The aircraft industry, for example, has been using them for elevated loads in structures (3b) for a long time due to their strength-to-weight ratio, for example, in skins, stringers, and spars[4].



(a) Plane structure (b) Stringers

Figure 3: Stringers [12].

In the wind energy sector, which is more modern, composite materials are essential for the viability of the structural design of the blades. However, composite materials can suffer significant damage along their life span. There are various reasons, such as lightning [5], bird strikes, hail, and high-intensity fatigue loads. If the damage is severe enough, it will affect the structure's performance; as a result, there is a need to repair these composite materials.

1.3 Recycling wind turbines

A significant number of parts can be recycled from wind turbines, such as the foundation, tower, gearbox, and generator[14]. However, the blades of all these wind turbines will have to be recycled or disposed of. This might become an issue because the blades are made of composite materials, namely fiberglass, which is challenging to recycle. Fixing and repairing wind turbine blades might be an excellent option to consider to extend their life cycle. Afterward, the best option to recycle the blade right now is making cement for construction sites [11]. Cement produces large amounts of CO2, and, as a result, it is not the greenest recycling method. To summarize, it is of vital importance to extend the life cycle of the blades because right now, the recycling alternatives are not good enough.

1.4 Types of repairs in composite materials

As mentioned before, various external variables, such as harsh meteorology or fatigue loads, can negatively affect wind turbine blades during their life cycle. Blades are forced to withstand difficult conditions, and as a result, the blades progressively begin to have more defects, such as cracks or leading edge erosion. These blades must be repaired, so they don't fail prematurely, shortening their life span.[13]

There are numerous ways of fixing composite materials, for instance, mechanical fastening, adhesive bonding, or hybrid fastening and bonding [4]. Mechanical fastening can be a problem due to the increase in weight and loss of aerodynamic performance, and therefore, bonded repairs are commonly preferred [4].

The desire to obtain a repair with minor peeling in the adhesive layer and that can withstand a high shear stress environment [20] has led the scientific community to focus on scarf repairs, ramped scarf repairs with small angles[20], and step scarf repairs. Figure 4. Nevertheless, even if it has been demonstrated that the repair can regain the static strength [9]. The area of the patch can become considerably much bigger than the defect; hence, the design needs to be optimized.

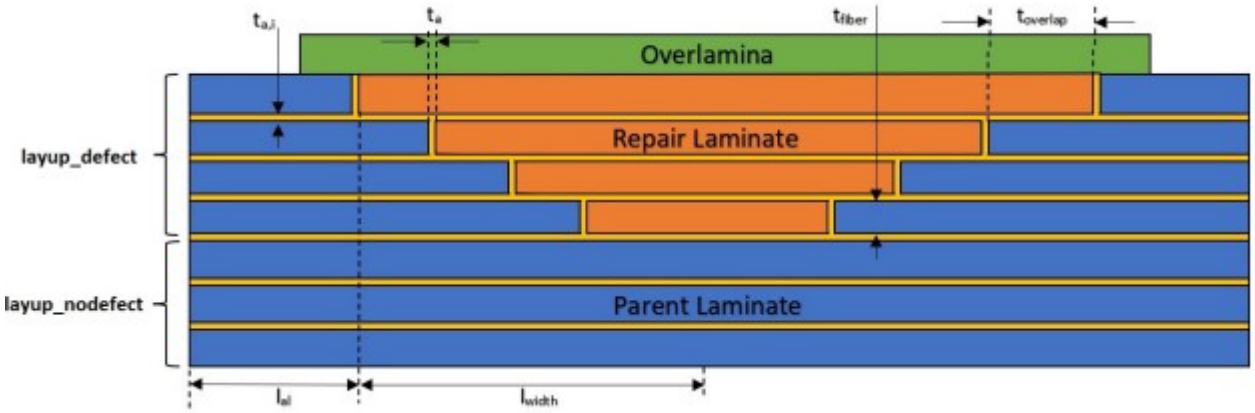


Figure 4: Step scarf repair scheme [13]

1.5 Project focus

This project aims to help scientists study the step scarf repair in the future. If, for example, an engineer wanted to simulate a step scarf repair, the engineer would need to simulate the model from scratch each time. If the engineer would like to optimize the repair, it would take a vast amount of time to keep changing the parameters of the laminate to get the optimum repair. To summarise, the MPSS allows an engineer to solve the time-consuming problem of creating the .inp file each time because it creates the .inp file for the engineer automatically.

This project has been created by improving and using an automated model generation tool called Multi-Parameter Stepped Scarf or MPSS[13]. The MPSS algorithm allows the user to

create a composite laminate input file that can be used in the FE program called ABAQUS.

The main contribution to the development of this software has been the creation of a graphical user interface or GUI and adding the cohesive resin option to the algorithm. The GUI is designed to be a user-friendly software for scientists using the MPSS algorithm. The GUI will allow the scientists to use the MPSS without searching and understanding all the variables in an extensive code. As explained earlier, the MPSS creates an input laminate file for the user without having to model it in a FE program.

Furthermore, the MPSS has been linked with ABAQUS so that the user does not have to load and simulate the laminate each time they design a laminate with the MPSS. The results are also read and displayed in the GUI.

Finally, in the results in section 5, the strength of scarf-repaired composite laminates is examined using the GUI to show its usefulness. First, the two FE elements used for the resin bond layer are validated. The validation results are checked by comparing them with the results obtained by other researchers in their scientific papers.

In addition, the study is focused on one particular case to study the failure process in more detail.

2 Algorithm concept

The GUI connects three environments, the GUI, MATLAB, and ABAQUS. First, the GUI is the interface from which the user inputs the desired laminate geometry and properties. The GUI converts the input and sends the information to the MATLAB functions. The MATLAB code creates a repaired laminate input file which is then sent to ABAQUS. ABAQUS is the FE program that simulates the laminate with the desired load. It is important to remember that FE programs approximate natural physical phenomena and should not be trusted entirely because there can be some errors that can modify the results. The errors can come from different sources such as:

- **Real system**:
  - Simplification of shape
  - Homogeneous material assumption
  - Simplification of contitutive law
  - Negligence of time dependence
  - Negligence of physics
  - Simplification from 3D to 2D
  - Loading situations
  - Boundary conditions
- **Physical model**:
  - Choice of element approximation
  - Choice of mesh
  - Approximation of boundaries
  - Numerical integration
- **Mathematical model**
  - Programming errors
  - Errors in input
  - Truncation errors (too few digits)
  - Round errors
- **Computer program**:
  - Interpretation of results

Once the simulation is over, the GUI reads the results and shows them in: the Run/results window and the Output: Stress & Strain window of the GUI.

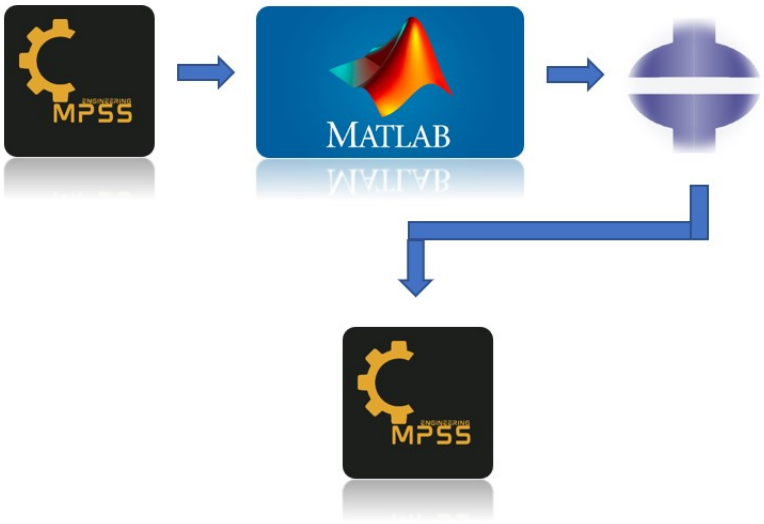


Figure 5: Concept

This GUI allows a future user to efficiently study repaired composite laminates with different material types, dimensions, loads, and other conditions, than those used in the study and run them automatically in ABAQUS. The GUI is detailed in the GUI documentation in section 7. To summarize, the user can view in the GUI the following: the creation of the repair laminate input file step by step, the load vs. displacement curve until critical failure if the load is high enough, and the stiffness of the laminate after the reapir. Additionally, the user can view the maximum strain and stress of each type of element and the reaction forces of the laminate. The code also creates a .txt file with the results, in case the user wants to plot the results elsewhere.

3 GUI development

The GUI has been developed using the MATLAB environment Appdesigner. The GUI has been designed with eight different windows. Four of the eight windows are for input parameters and material properties, and 2 for running the code and showing the simulation results. Finally, the last 2 are user guides of the different MPSS parameters that the algorithm uses. The idea behind having a small user guide of only the MPSS properties is that a new user can learn the meaning of the variables without having to search in the GUI documentation (Appendix A section 7).

The 4 input windows are:

- General input
- Geometry
- Material properties
- Cohesive material properties & General Static Analysis & Step control

The 2 output windows are:

- Run/Output
- Output: Stress & Strain

The last two windows are the user guides:

- Guide 1
- Guide 2

3.1 General input

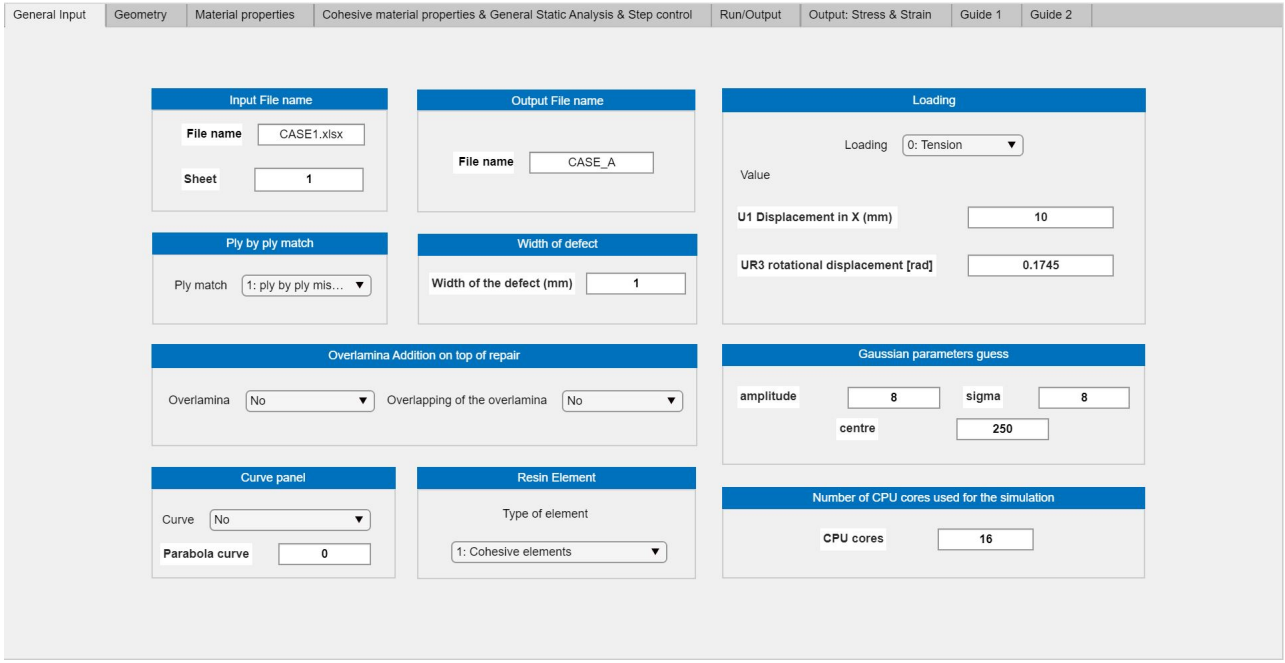


Figure 6: General input

In General input Figure 51 the following components have been added:

- **Input File name**: this text edit field has been designed to allow the user to input the name of the wrinkled file. The program will read it if it is in the same folder as the program.
- **Output File name**: this text edit field has been designed to name the files MPSS and ABAQUS will create. This includes the name of .inp, .odb, .mdl, .sim, .prt, .msg, .com, and .dat files.
- **Ply by ply match**: this drop-down aim is to give the user the option of having a mismatch in the laminate.
- **Loading**: this box has been designed to input the load that the laminate will be submitted to in the form of displacement/rotation during the simulation. The drop-down allows the user to change between tension or bending and the numeric edit fields to input the desired load.
- **Overlamina Addition on top of the repair**: these two drop-downs have been designed to let the user choose between having an overlamina on top of the repair and if that overlamina also has an overlapping layer length.
- **Gaussian parameters**: these three different numeric edit fields permit the user to input the gaussian interpolation parameters that affect the interpolation curve.

- **Curve panel**: this box is designed to allow the user to curve the laminate in case they want to. The drop-down lets the user choose to curve it or not, and the numeric field input the desired curvature.
- **Number of CPU cores**: this numeric edit field is designed to let the user choose the number of cores that will carry out the simulation in ABAQUS.

3.2 Geometry

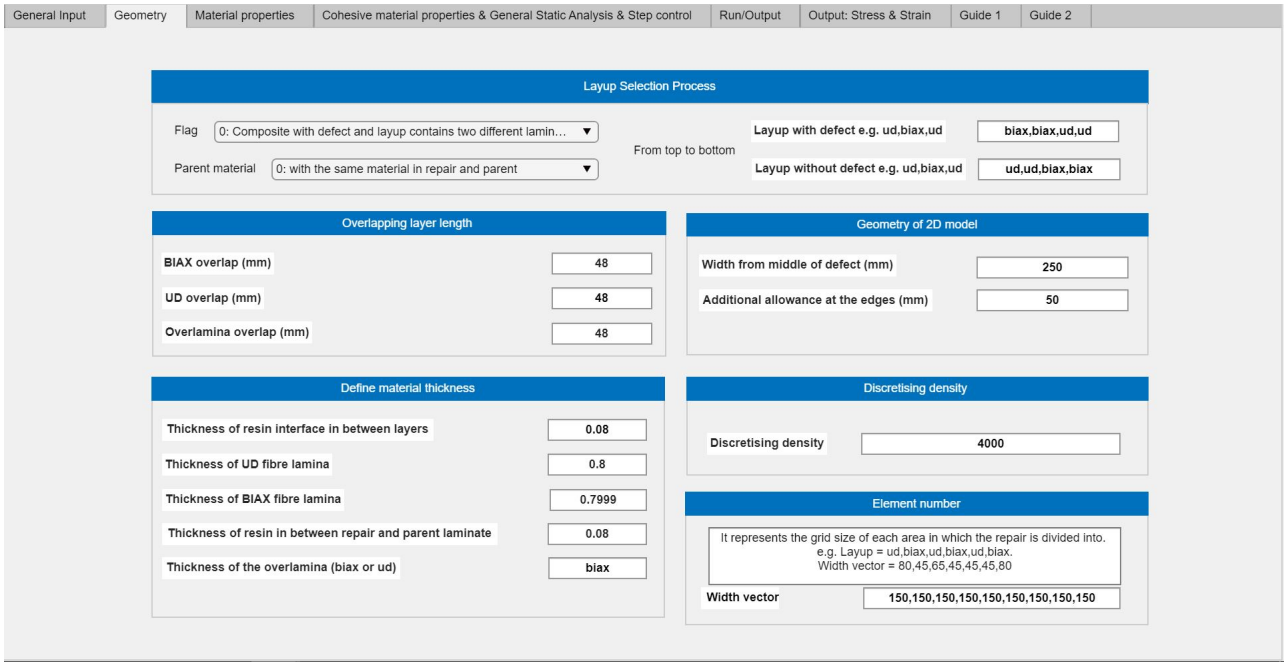


Figure 7: Geometry

In Geometry Figure 52, the following components have been added:

- **Layup selection process**: this box is significant because it lets the user choose the type function that will be used to create the .inp file and the layup of the laminate.
  - **The flag drop-down** is designed to let the user choose between the pristine option, which creates a non-repaired laminate and a repaired laminate. This drop-down also permits the user to choose if the layup has 1 or 2 material types. A tool tip pops up when the mouse is on top of the flag drop-down to check that the user uses configurations that match.
  - **The parent drop-down** is designed to give the user the option to change the UD/BIAX material of the parent and the repair so that even if both parent and repair are UD, they won't have the same properties.
  - **Layup with and without defect**: these variables represent the repaired portion of the laminate and the parent portion, respectively. 2 tooltips that pop up when

the mouse is on top of the text edit fields have been added to explain that the plys will be added from top to bottom. Both text edit fields create a vector that is compared element by element to create the laminate ply vector from top to bottom.

- **Overlapping layer length**: this box contains three numeric edit fields that are designed to let the user input the numeric values of the three types of overlapping layer lengths: UD, BIAX, and Overlamina.
- **Geometry of the 2D model**: these two numeric edit fields have been added so that the user can input the desired dimensions of the laminate.
- **Define material thickness**: these four numeric edit fields allow the user to input the material thickness. A tooltip has been added to check that the user does not input the same exact number for UD and BIAX because, in that case, the algorithm fails. The overlamina thickness text edit field lets the user select the material used for the overlamina.
- **Discretasing density**: this numeric edit field is designed to let the user select the mesh density of the laminate. (section 7)
- **Width vector**: This text field is designed to read a numeric vector with elements separated by commas and then convert it into an array for the code. It represents the number of elements in each section of the repair. (section 7)

3.3 Material properties

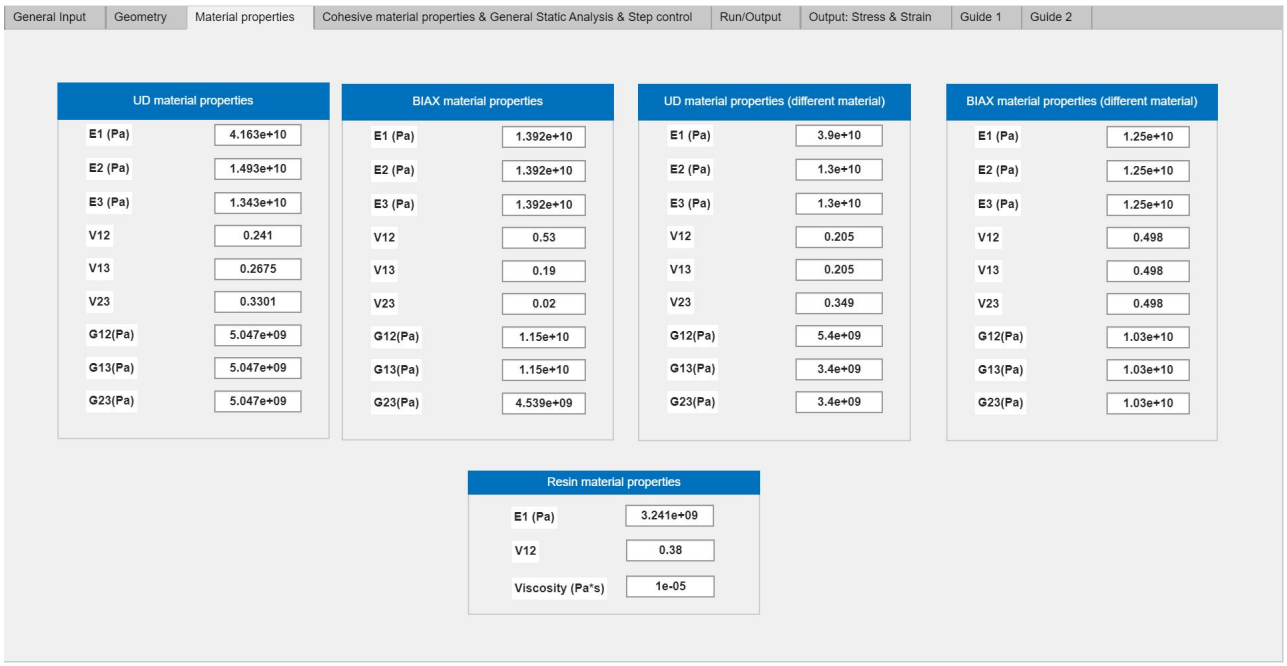


Figure 8: Material properties

In the material properties window Figure 8, the following components have been added:

- **UD material properties**
- **BIAX material properties**
- **UD material properties** (different material): these properties are only used in case the parent and the repaired material are not identical.
- **BIAX material properties** (different material): these properties are only used in case the parent and the repaired material are not identical.
- **Resin material properties**

All these properties are designed as numeric edit fields that read the input from the user.

3.4 Cohesive material properties & General Static Analysis & Step control

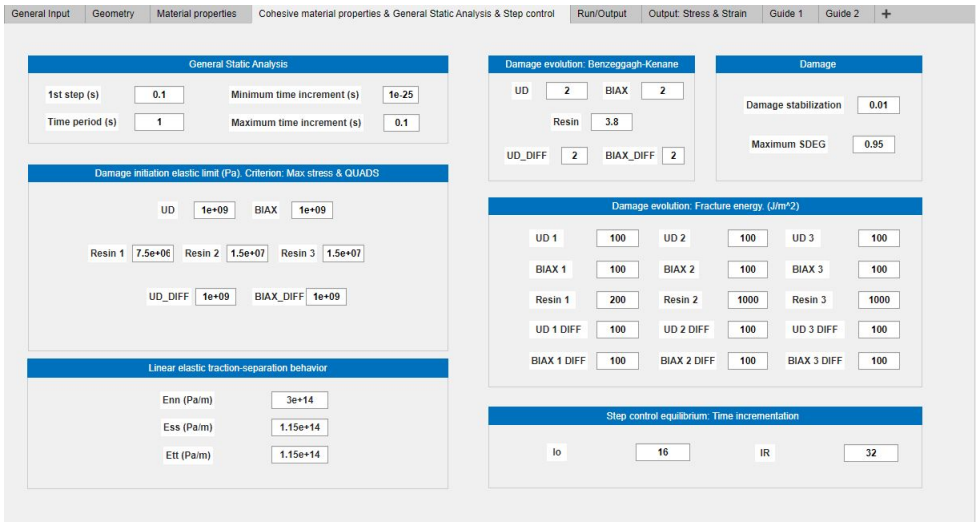


Figure 9: Cohesive material properties, General Static Analysis and Step control

In Cohesive material properties & General Static Analysis & Step control window Figure 9, the following components have been added:

- **General static analysis**: this box's numeric edit fields are designed to enable the user to input the static analysis variables used in the ABAQUS simulation.
- **Damage initiation elastic limit (Pa)**. Criterion: MAXS & QUADS: this box includes the numeric edit fields that will be used in the ABAQUS simulation for the XFEM damage initiation in the case the UD/BIAX elements reach their elastic limit. If the cohesive option has been chosen for the resin elements in the GUI, this value will represent the elastic limit for the resin. (section 7)

- **Elastic traction separation behavior**: This box includes three numeric edit fields that enable the user to input the initial elastic stiffnesses of the traction separation law: E_{nn}, E_{ss}, and E_{tt}.
- **Damage evolution: Benzeggagh-Kenane**: this box includes five numeric edit fields designed to let the user introduce the power factor in the Benzeggagh-Kenane damage approach.
- **Damage**: this box includes two numeric edit fields. Damage stabilization allows the user to input the desired damage stabilization for the XFEM. Maximum SDEG allows the user to input the stiffness degradation limit at which ABAQUS will erase the elements.
- **Damage evolution**: this box includes three numeric edit fields for each material type. The numeric fields are designed to let the user input the fracture energy at which the material will reach maximum damage and break.
- **Step equilibrium & time incrementation**: In this box, two numeric edit fields are used to enable the user to input the convergence parameters I_o (number of equilibrium iterations after which the check is made that the residuals are not increasing in two consecutive iterations) and I_R (number of equilibrium iterations after which the logarithmic rate of convergence check begins).

If the cohesive option element has been selected, the cohesive properties will be used in the simulation.

3.5 Run/Output

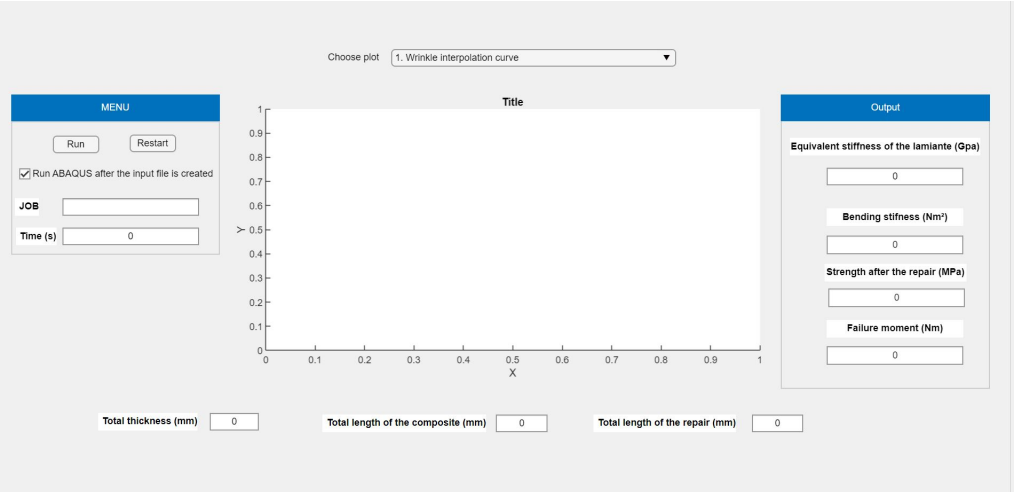


Figure 10: Run and Output

For the Run/Output window Figure 10 the following components have been added:

- **Menu**: in this box, two different push buttons have been added:
  - **Run**: When this button is pushed, the algorithm is run. The button has a callback function that includes the whole MPSS, the ABAQUS sending, and reading functions, and the code needed to calculate and show the results in the Run/Output window.
  - **Restart**: when this button is pushed, a callback function clears the plots and all the output numeric and text files.
  - **Job**: this text edit field is designed to let the user know the status of the running algorithm. If there is an error in the MPSS, it will show Error. When the .inp file is sent to ABAQUS, it will change to the state Running ABAQUS, and when the results have been read and the stiffness calculated, it will change to the state Done.
  - **Time**: This numeric edit field will show the amount of time the program has taken to run the algorithm completely when it is over.
  - **E**: This numeric edit field is calculated after the ABAQUS results are read in case a tension load has been applied. The number is calculated with the function Youngs_modulus.m. (Appendix B Equation 11)
  - **D**: This numeric edit field is calculated after the ABAQUS results are read when a bending load has been applied. The number is calculated with the function Bending_stiffness.m. (Appendix B Equation 14)
- **Total thickness**: this numeric field shows the resulting final thickness of the laminate when the MPSS algorithm is completed.
- **Total length**: this numeric field shows the final length of the laminate when the MPSS algorithm is completed.
- **Total length of the composite**: this numeric field shows the final length of the laminate when the MPSS algorithm is completed.
- **Total length of the repair**: this numeric field shows the resulting repaired patch length of the laminate when the MPSS algorithm is completed.
- **Choose plot**: this drop-down is assigned to a callback function that plots the different steps while creating the repaired laminate. It also can show a load vs. displacement graph of the laminate.

3.6 Output: Stress and Strain

Figure 11: Output Stress and Strain

For the Output: Stress and Strain window Figure 11, the following components have been added:

- **Choose element type**: this drop-down is designed to show the maximum stress and strain depending on the element type selected. The drop-down has a callback function that reads the .dat file. This feature has been added to have a quick view of the stresses and strains without opening the ABAQUS .odb file.
- **Reaction forces**: these three numeric edit fields show the reaction forces of the last increment to the user once the simulation is over and the results are read.

3.7 Guide 1 and Guide 2

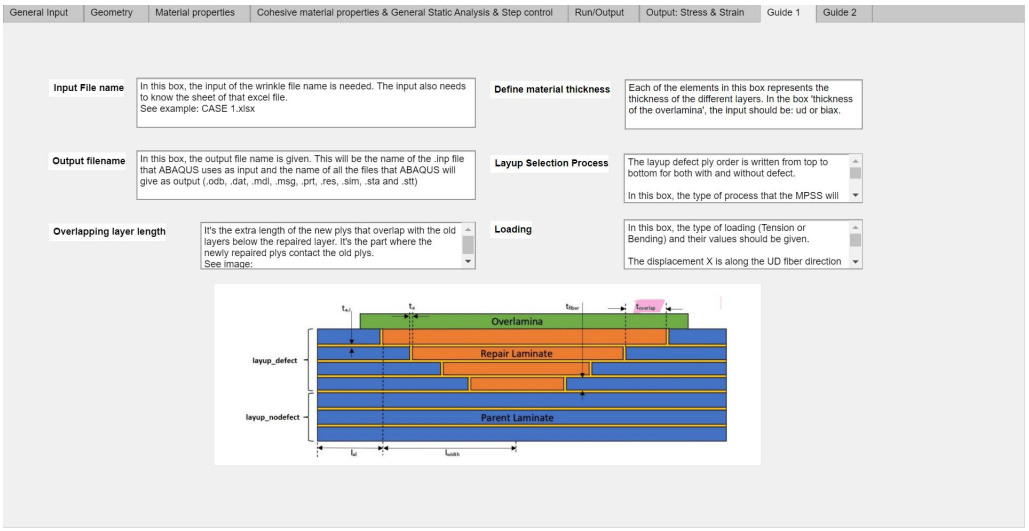


Figure 12: Guide1

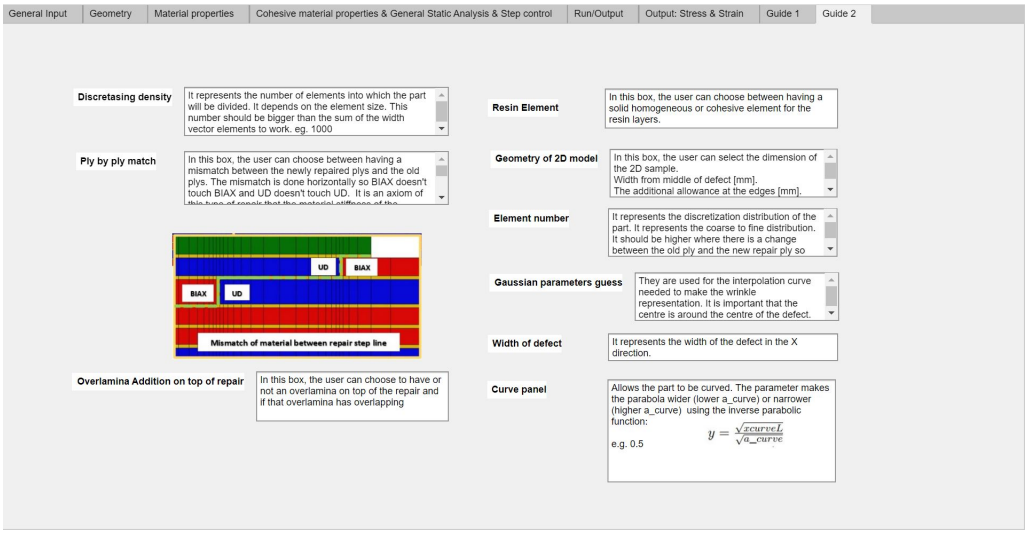


Figure 13: Guide2

The last 2 windows are Guide 1 (Figure 12) and Guide 2 (Figure 13). They have been added to help novice users understand the MPSS algorithm's variables. They are made with the label tool of the MPSS.

4 FE model

4.1 SI unit system

The model .inp file is created with the MPSS. The FE model uses the following unit system:

Quantity	SI unit
length	m
Force	N
Mass	kg
Time	s
Stress	Pa
Energy	J
Density	kg/m^3

Table 1: SI unit system

4.2 Model functions

The FE input file is created automatically from MATLAB at the end of the MPSS algorithm. Depending on which configuration the user has selected, there are 12 different functions that create an input file.

If the repair and parent material are the same as explained in section 3, there are eight functions that create the designed input file.

- – If the loading type is tension
  - ∗ If the element type for the resin is Homogeneous element or CPE4, then the function that creates the input file is K_abaqusrepairSolid_function.m. (Appendix C Figure 64)
  - ∗ If the element type is Cohesive element or COH2D4, then the function that creates the input file is K_abaqusrepairCZM_functionCPE4_2.m. (Appendix C Figure 64)
    Note: This function has been changed because it used element type COH2D4 for all the resin, including the resin pocket. Since the thickness of the resin pocket (in the case of a mismatch) is thick and cohesive elements work well for thin layers, not for thick layers, the resin pocket has been modeled with CPE4 elements instead of COH2D4.
- – If the loading type is bending
  - ∗ If the element type for the resin is Homogeneous element or CPE4, then the function that creates the input file is L_abaqusrepairSolid_function.m. (Appendix C Figure 64)

        * If the element type is Cohesive element or COH2D4 then the function that creates the input file is L_abaqusrepairCZM_function.m. (Appendix C Figure 64)

- If the laminate is a pristine laminate
  - If the loading type is tension
    * If the element type for the resin is Homogeneous element or CPE4, then the function that creates the input file is K_abaquspristineSolid_function.m. (Appendix C Figure 64)
    * If the element type is Cohesive element or COH2D4 then the function that creates the input file is K_abaquspristineCZM_function.m. (Appendix C Figure 64)
  - If the loading type is bending.
    * If the element type for the resin is Homogeneous element or CPE4, then the function that creates the input file is L_abaquspristineSolid_function.m. (Appendix C Figure 64)
    * If the element type is Cohesive element or COH2D4 then the function that creates the input file is L_abaquspristineCZM_function.m. (Appendix C Figure 64)

If the repair and parent materials are not identical as explained in Figure 3.2.

- If the loading type is tension
  - If the element type for the resin is Homogeneous element or CPE4, then the function that creates the input file is K_abaqusmatdiffSolid_function.m. (Appendix C Figure 64)
  - If the element type is Cohesive element or COH2D4 then the function that creates the input file is K_abaqusmatdiffCZM_function.m. (Appendix C Figure 64)
- If the loading type is bending
  - If the element type for the resin is Homogeneous element or CPE4, then the function that creates the input file is L_abaqusrepairSolid_function.m. (Appendix C Figure 64)
  - If the element type is Cohesive element or COH2D4 then the function that creates the input file is L_abaqusmatdiffCZM_function.m. (Appendix C Figure 64)

To check the code of the functions go to the repository Appendix C Figure 64

4.3 Material properties

The following material properties from [1] have been used for the GUI testing:

Material	Property	Value	Unit
UD [0°]	E_11	41.63	GPa
	E_22	14.93	GPa
	E_33	13.43	GPa
	V_12	0.241	-
	V_13	0.2675	-
	V_23	0.3301	-
	G_12	5.047	GPa
	G_23	5.047	GPa
	G_13	5.047	GPa
BIAX [±45°]	E_11	13.92	GPa
	E_22	13.92	GPa
	E_33	13.92	GPa
	V_12	0.53	-
	V_13	0.19	-
	V_23	0.02	-
	G_12	11.5	GPa
	G_23	11.5	GPa
	G_13	4.539	GPa
Resin [-]	E_11	3.241	GPa
	v_12	0.38	-
	G_12	1.17	GPa

Table 2: Material properties

The following cohesive material properties from [1] have been used for the GUI testing Figure 54:

Material	Property	Value	Unit
Epoxy resin	K_I	$3 \cdot 10^{14}$	N / m^3
	K_I I	$1.15 \cdot 10^{14}$	N / m^3
	K_I I I	$1.15 \cdot 10^{14}$	N / m^3
	σ_{ult_I}	7.5	MPa
	$\sigma_{ult_{II}}$	15	MPa
	$\sigma_{ult_{III}}$	15	MPa
	G_I c	200	J / m^2
	G_I I c	1000	J / m^2
	G_I I I c	1000	J / m^2

Table 3: Cohesive material properties

4.4 Loading and boundary conditions

There are two types of boundary conditions depending on the load: tension or bending. For both cases, two master nodes were used to control the node sets. There are two node sets called REPAIREDPANEL-1.LEFTEDGENODESET and REPAIREDPANEL- 1.RIGHT-EDGENODESET. The node sets represent the nodes on the edges left and right of the laminate, respectively.

In the case of tension, the boundary conditions are illustrated in Figure 60. On the left edge of the laminate, the master node RP_LEFT, also called RP-1, is completely constrained. The two displacement directions, which correspond to the displacement in the X direction and the Y direction, respectively, are U1 = 0 and U2 = 0. The rotational degree of freedom in the Z direction, which corresponds to UR3, is UR3 = 0; thus, the displacement/rotation in all 2D directions is constrained to 0. However, on the right edge, the master node RP_RIGHT, also called RP-2, is not completely constrained U1 $\neq$ 0 [m]. This non-zero displacement is the load that the laminate is subjected to. The other two degrees of freedom are fixed on the right edge, U2 = 0 and UR3 = 0.

In the case of bending, the boundary conditions are illustrated in Figure 61. The laminate constraints are the same as a simply supported beam. On the left edge, RP_LEFT, and the right edge, or RP_RIGHT, the two displacement degrees of freedom are U1 = 0 and U2 = 0. The rotational degrees of freedom in the Z direction on both nodes are UR3 $\neq$ 0. These non-zero rotations are the bending force that the laminate is subjected to. On RP_LEFT UR3 = $-\theta$ [rad], and on RP_RIGHT UR3 = θ. The angle θ is measured counter-clockwise.

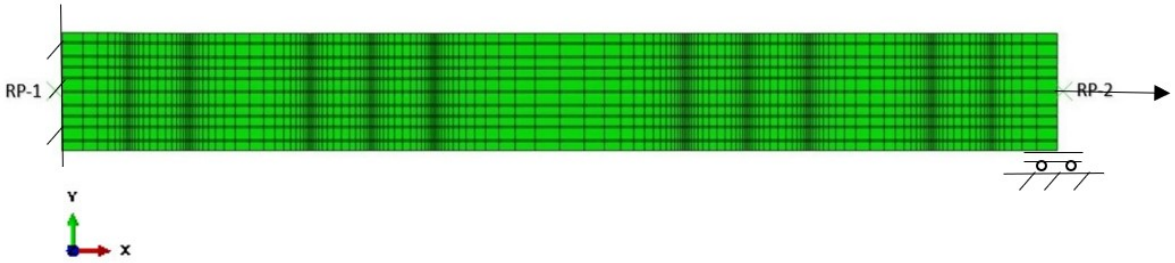


Figure 14: BC for tension

Figure 15: BC for bending

4.5 Element types

The main element type used in the model is CPE4 (4-node bilinear). This element type affects the following element sets:

- EOVERLAM: this element set includes the overlamina elements, in case the overlamina option has been selected as 'Yes'. Figure 17
- EPARENTBIAX: this element set includes all the elements in BIAX plies that the parent laminate may have. Figure 18
- EPARENTUD: this element set includes all the elements in UD plies that the parent laminate may have. Figure 18
- EREPAIRBIAX: this element set includes all the elements in BIAX plies, in case the mismatch option has been selected as 'No'.Figure 18
- EREPAIRUD: this element set includes all the elements in UD plies, in case the mismatch option has been selected as 'No'.Figure 18
- EREPAIRBIAX1: this element set includes all the elements in BIAX plies, in case the mismatch option has been selected as 'Yes'.Figure 18
- EREPAIRUD1: this element set includes all the elements in UD plies, in case the mismatch option has been selected as 'Yes'.Figure 18
- ECOMPOSITEBIAX: this element set includes all the elements in BIAX plies, in case the pristine option is selected in FLAG as two or 3.Figure 16
- ECOMPOSITEUD: this element set includes all the elements in BIAX plies, in case the pristine option is selected in FLAG as two or 3.Figure 18

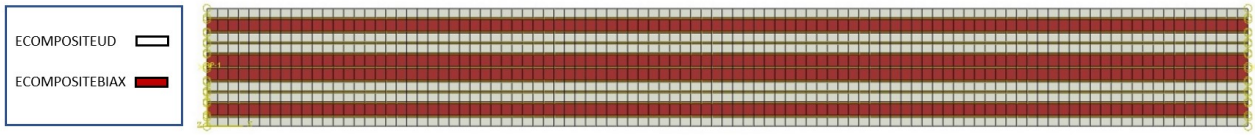


Figure 16: Element sets in a pristine model [13]

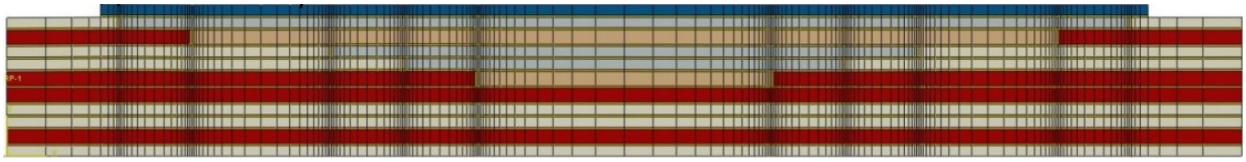


Figure 17: Element sets with overlamina [13]

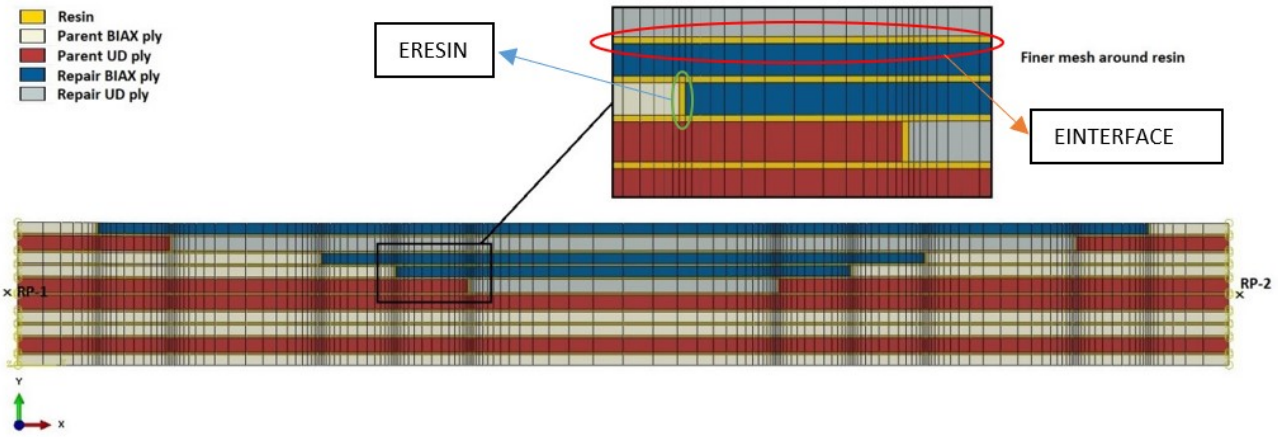


Figure 18: Element sets [13]

In the MPSS the user can choose between 2 types of elements for the resin in the laminate:

- 2D solid elements, which are CPE4 (4-node bilinear).
- 2D Cohesive elementes, which are COH2D4 (4-node two-dimensional cohesive element)

These element types affect the following element sets:

- EINTERFACE: this element set includes the horizontal resin layer between the plies.
- ERESIN: this element set includes the vertical resin elements between parent and repair.

In case the mismatch option has been selected as 'Yes', the resin pocket at the bottom of the repair Figure 19 is treated as a CPE4 because cohesive elements work best when their thickness is small compared to their length. This element set is called ERESINRICH.

Figure 19: Element set with mismatch [4]

4.6 Numerical set-up and computer information

The discretasing density can be changed in the GUI. This allows the user to get a more precise mesh with more elements if needed. The cases were run with a discretasing density of 4000. Depending on the case the number of elements and nodes varied. For the main cohesive case of $\beta = 1/60$ explained in section 5 the total number of nodes was 38544 and the total number of elements was 36120. The minimum size of and element was 0.0252 mm in width and 0.2 mm in height, which corresponds with the thickness of the resin layer.

The model was run in the DTU linux Citrix server cluster. The server uses 6 computer nodes which are E5-2667 v3 with 8 cores of 3.2-3.6 GHz. The graphics and data processing is E5-2667 v3 with 8 cores and 2xNVIDEA Quadn K4200. The simulations took up to 30 minutes for the homogeneous resin option, and up to 16 hours for the cohesive resin element option.

5 Results

5.1 Validation of the homogeneous resin element model (CPE4)

5.1.1 Laminate geometry

First, validating the model is necessary. The results have been compared with the following previous papers: [21] for the resin CPE4 model and [4] for the resin COH2D4 model. The homogeneous resin element option is studied in this section. The graphs explained in this section correspond to the scarfed resin path. Figure 20 represents the step scarf path; it does not correspond to the actual layup.

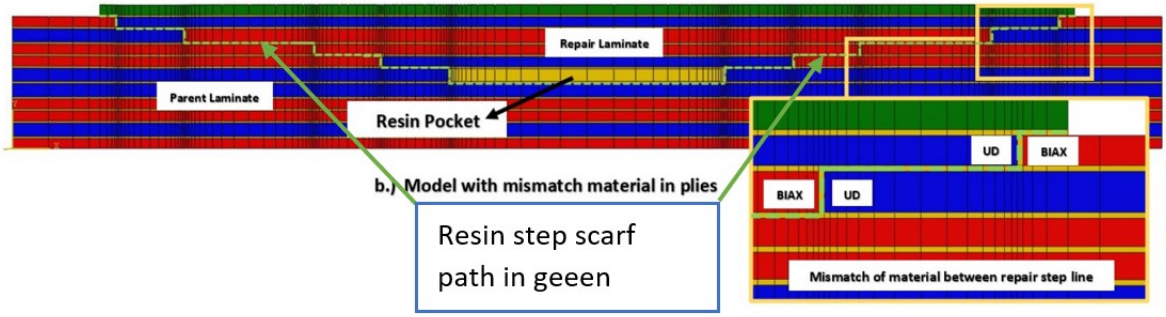


Figure 20: Step scarf path [13]

The mismatch option was selected and the geometry was the following:

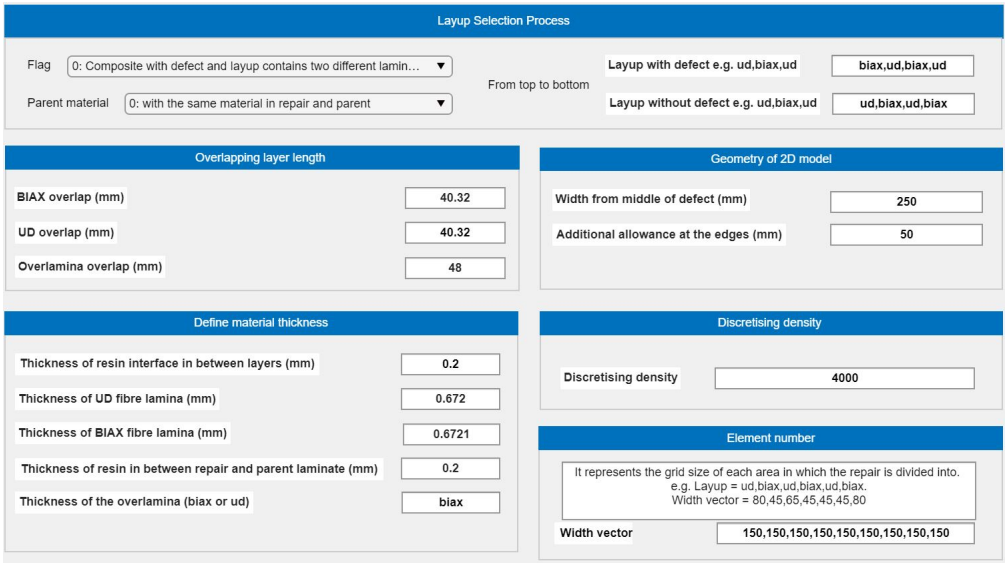


Figure 21: Geometry for the homogeneous validation

The laminate was subjected to a normal force produced by a displacement of the RP_RIGHT master node of 10 mm in the X direction.

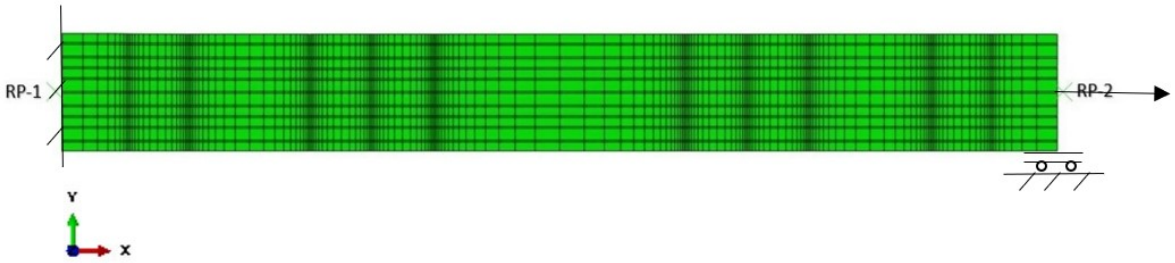


Figure 22: BC for the homogeneous element model validation

5.1.2 Results validation

The principal stresses along the scarfed resin path are shown in Figure 24. The maximum the stress values are reached in the vertical resin elements.

As shown in Figure 23 the vertical resin elements have a higher stress that the resin elements around them.

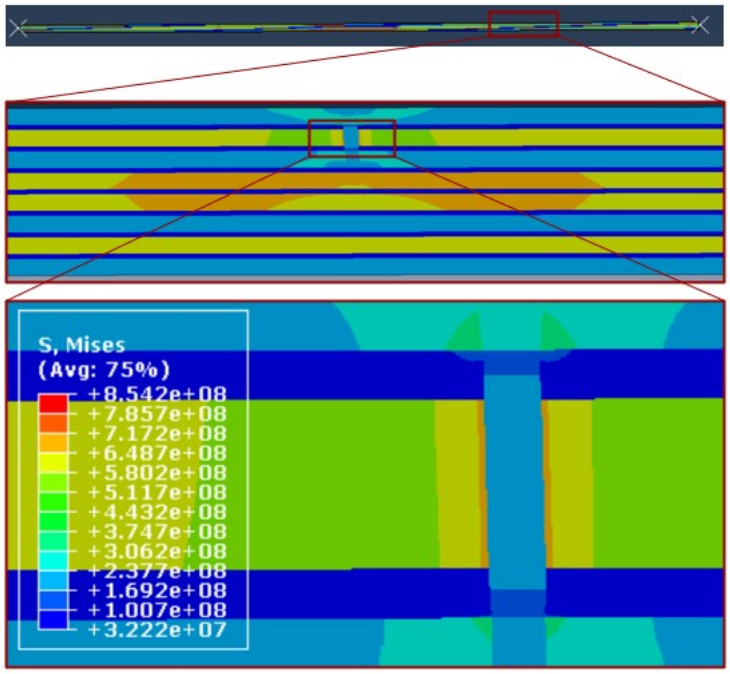


Figure 23: Stress in vertical resin elements from the GUI odb results

The resin element showed in Figure 23 corresponds with the second peak from the left Figure 24. All the other peaks correspond to the other vertical resin elements in the laminate. The plot symmetry is caused by the symmetry of the laminate repair.

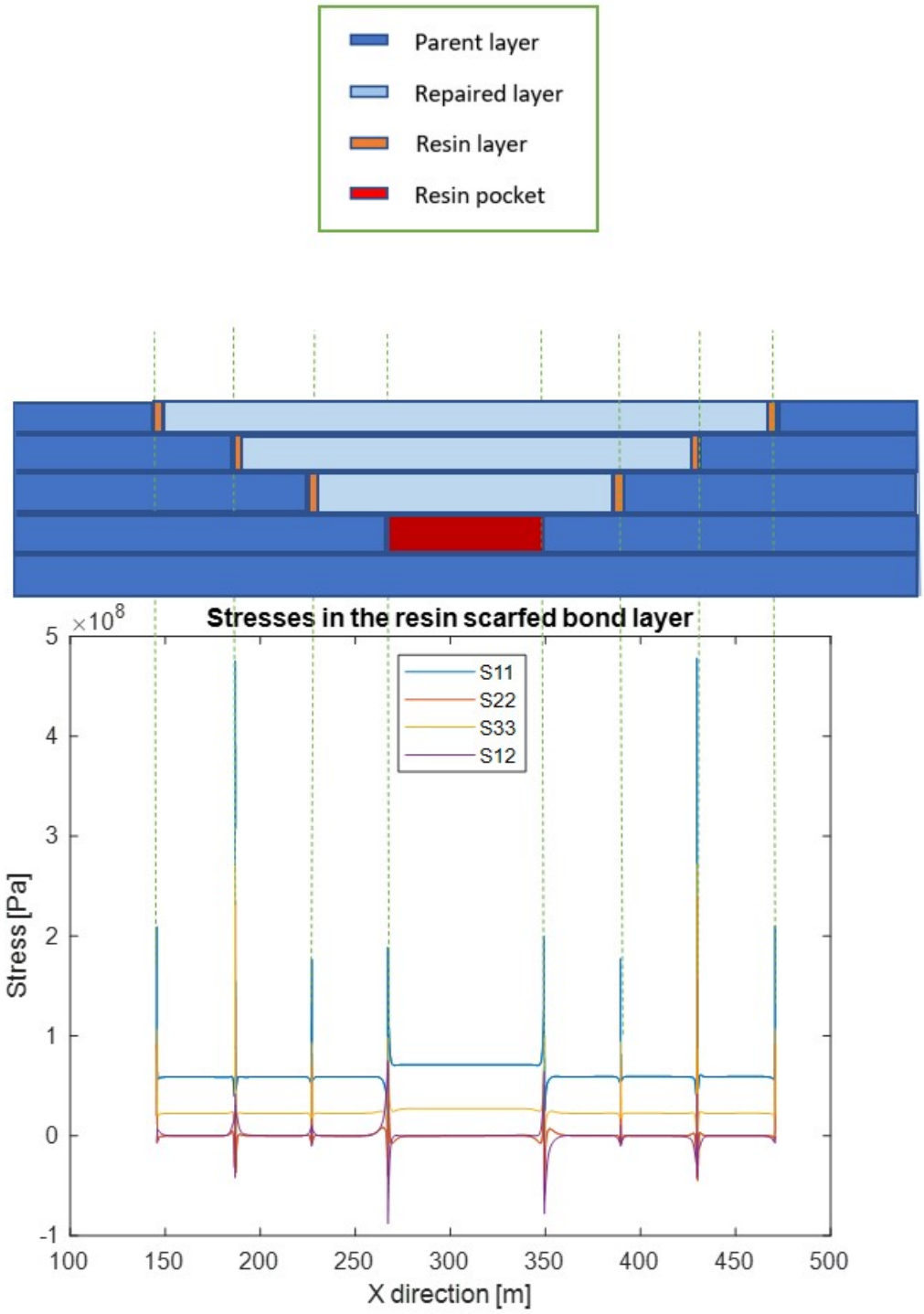


Figure 24: Stresses along the resin path

To validate the homogeneous model, the results are compared with the results of [21]. In their article they studied the failure mechanisms in stepped bonded structures using FEM

(Finite element method), experimental studies, and CDM (Continuum Damage Mechanics). In [21] Qing Han, Bin Liu & Wei Xie reached the following conclusions:

- Stress concentration locates at the vertical zone of adhesive.
- S11, S22 and S33 is greater than S12, S23 and S13.
- Among S11, S22, S33, S12, S23 and S13, S11 is maximum.

Firstly, the stress concentration of the resin bond locates at the vertical zone as shown in Figure 23 and Figure 24. Secondly, S11, S22 and S33 are higher than S12 as shown in Figure 24. Finally, S11 is the highest stress among S11, S22, S33 and S12 as shown in Figure 24. All of their results are achieved by the homogeneous resin element model (CPE4), hence, the model is validated.

5.2 Validation of the cohesive resin model (COH2D4)

5.2.1 Laminate geometry

For this validation, the geometry, boundary conditions and mismatch are the same as in the validation of the homogeneous element. The material properties used are explained in section 4.

5.2.2 β parameter theoretical explanation

β is an important variable used in the aerospace industry [4]. In this section β is studied to validate the model comparing it to the results in [4].
Beta is the ratio between the ply thickness and the overlapping layer length.

$$\beta = \frac{t}{OL}$$

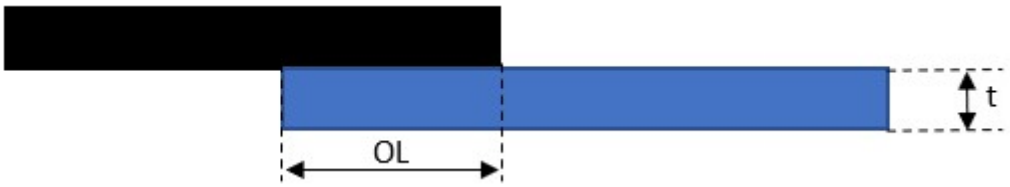


Figure 25: β geometry

β is an important parameter because it affects the load transmission along the laminate. The contact surface between the repair plies and the old plies is the point where the load carried by the parent fibers changes ply to the fibers of the repaired plies. The larger the bond between the two bonded parts, the better the load transmission will be. The resin has to hold the shear stress between the two bonded parts, and as a result, it is the place where the resin stress concentrates.

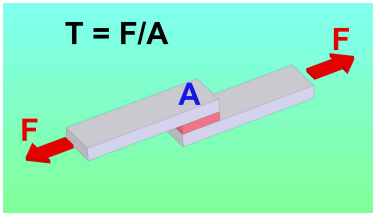


Figure 26: Shear between two bonded plies by resin [6]

5.2.3 Failure process

In this section, the failure of the laminate with $\beta = 1/60$ is examined in detail. The load displacement curve is showed in Figure 27.

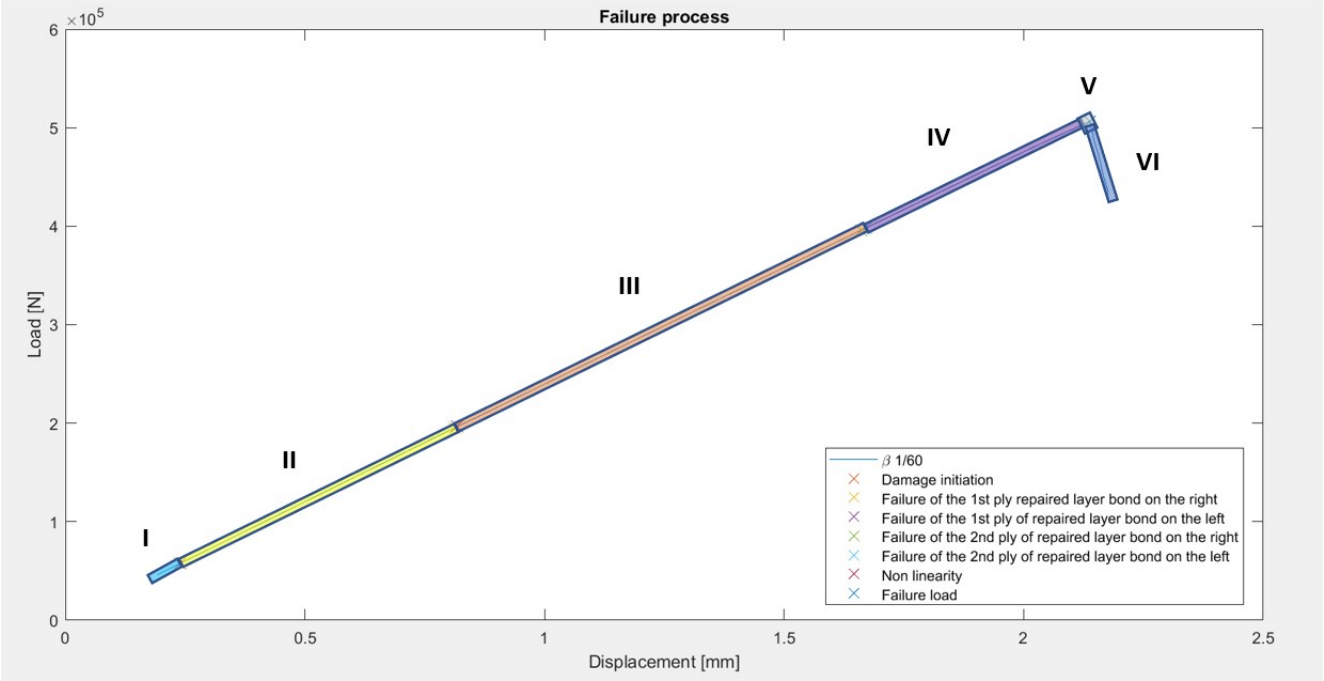


Figure 27: Failure mechanism

The failure mechanism is divided into 6 different stages:

- **I**: Before the plastic damage, the laminate elements show a elastic response, which corresponds to the elastic part of the traction separation law Figure 29. The stress concentration locates in the same places explained in the homogeneous validation.

- **II**: The first resin element reaches the damage initiation criteria limit, in this case, QUADS. The quadratic stress criteria or QUADS was chosen instead of the maximum stress or MAXS criteria.

$$QUADS\left\{\frac{\langle t_n\rangle}{t_n^o}\right\}^2+\left\{\frac{t_s}{t_s^o}\right\}^2+\left\{\frac{t_t}{t_t^o}\right\}^2=1 \tag{1}$$

$$MAXS\max\left\{\frac{\langle t_n\rangle}{t_n^o},\frac{t_s}{t_s^o},\frac{t_t}{t_t^o}\right\}=1 \tag{2}$$

The criteria QUADS was chosen because it is more restrictive than the MAXS criteria. For example, an element can reach the limit with $t_n/t_n^0 = 0.6$, $t_s/t_s^0 = 0.6$ and $t_t/t_t^0 = 0.6$ with the QUADS criteria and with the MAXS the limit it would not have been reached. The following image shows the first element to reach the limit in the laminate.

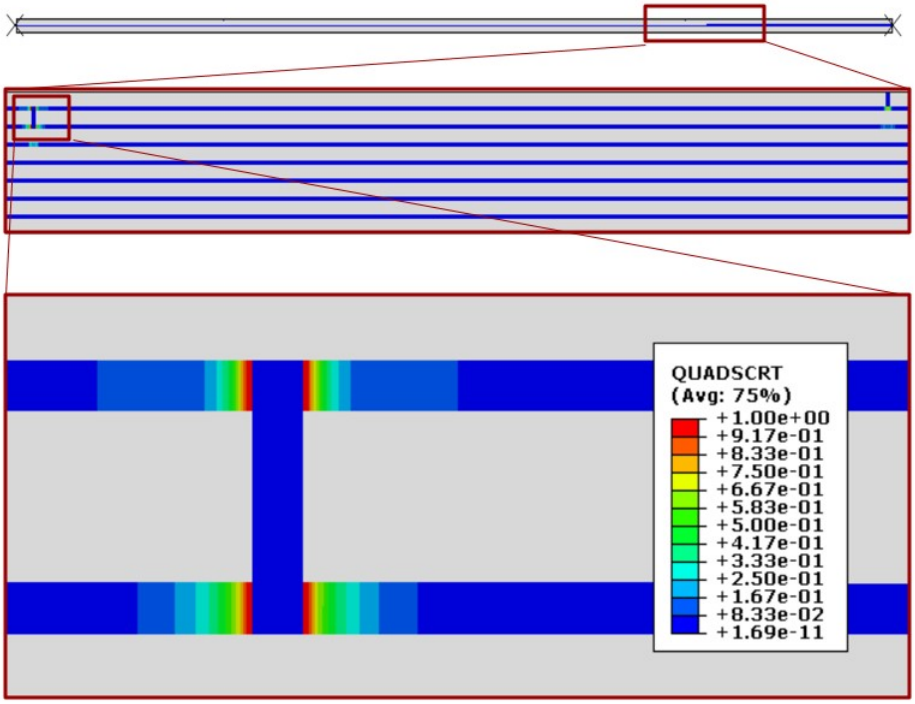


Figure 28: Damage initiation

During this stage II Figure 27 the damage in the resin elements starts to grow until the fracture energy limit is reached. At the start of this stage, the QUADS criteria limit is reached. Point A corresponds to the QUADS criteria limit in the traction separation law shown in Figure 29. During this stage the deformation in the elements is not longer elastic but plastic instead. The total energy under the triangle corresponds to the fracture energy. The damage evolution law used after the damage initiation limit

is Benzeggagh-Kenane, which is a mix mode energy based criteria.

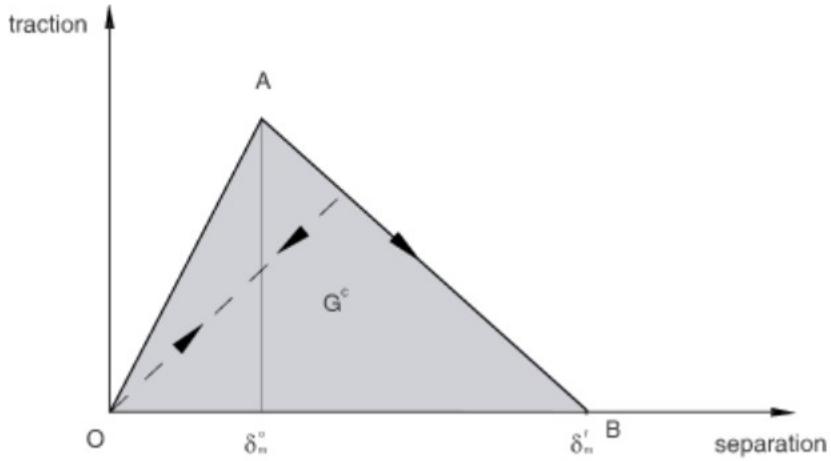


Figure 29: Traction separation law

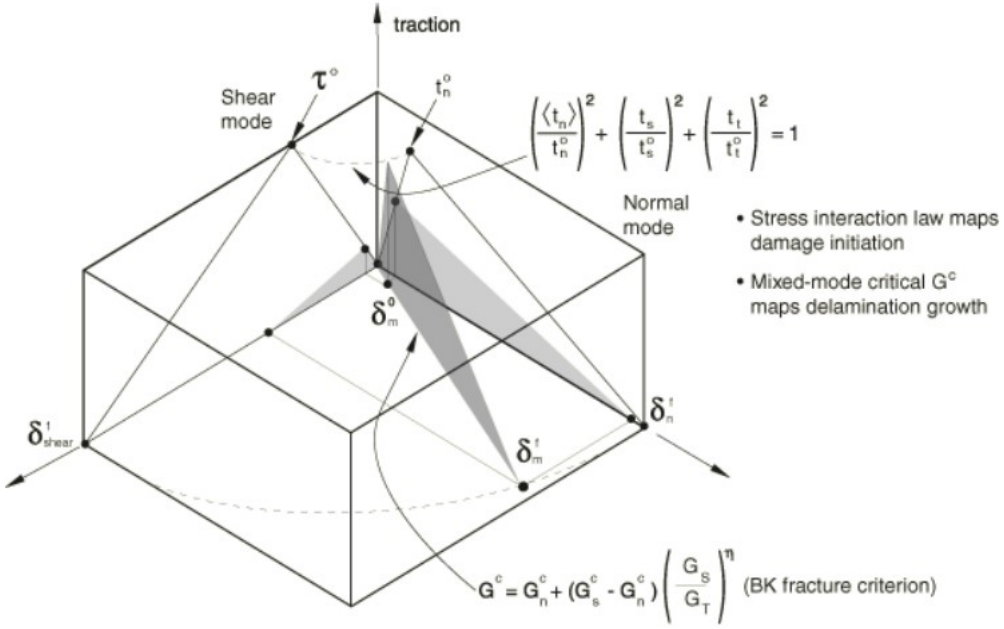


Figure 30: Mix mode response in cohesive elements

- **III**: The fracture energy limit G is reached in an element for the first time in the 1st resin layer. The maximum damage allowed was 0.95 before the element was deleted. The maximum damage was reduced from 1 to 0.95 due to convergence problems, therefore, the simulation considers the failure of the element at a damage of 0.95. In the ABAQUS view port the variable SDEG or stress degradation represents this damage, as shown in Figure 31

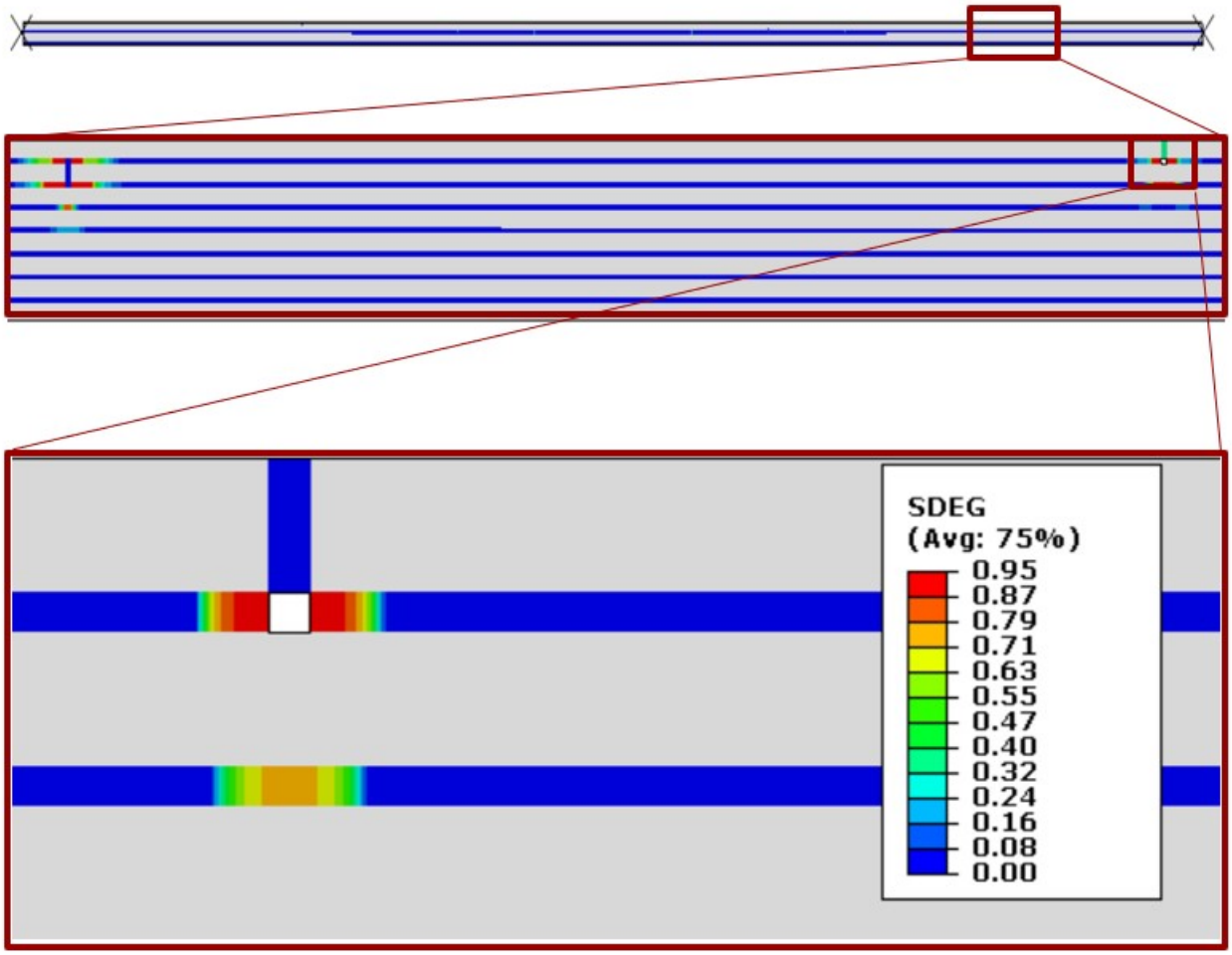


Figure 31: Crack appearance in the first step

During this stage III, the delamination of the first layer starts and the crack grows through the resin layer. The crack grows to the left where the second step is, and, it also grows to the right edge of the lamina. This delamination is caused by: the stress peel stress concentration (S22), and the interlaminar shear stress concentration (S12). This stresses are the cause of the failure of the laminate in Mode I and Mode II respectively. The different types of fracture modes are shown in Figure 32

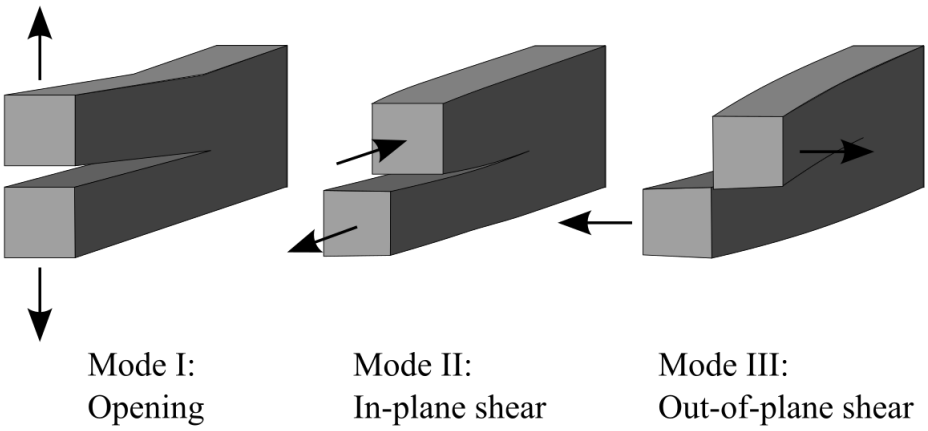


Figure 32: Fracture modes

The peel stress S22 is shown in Figure 33

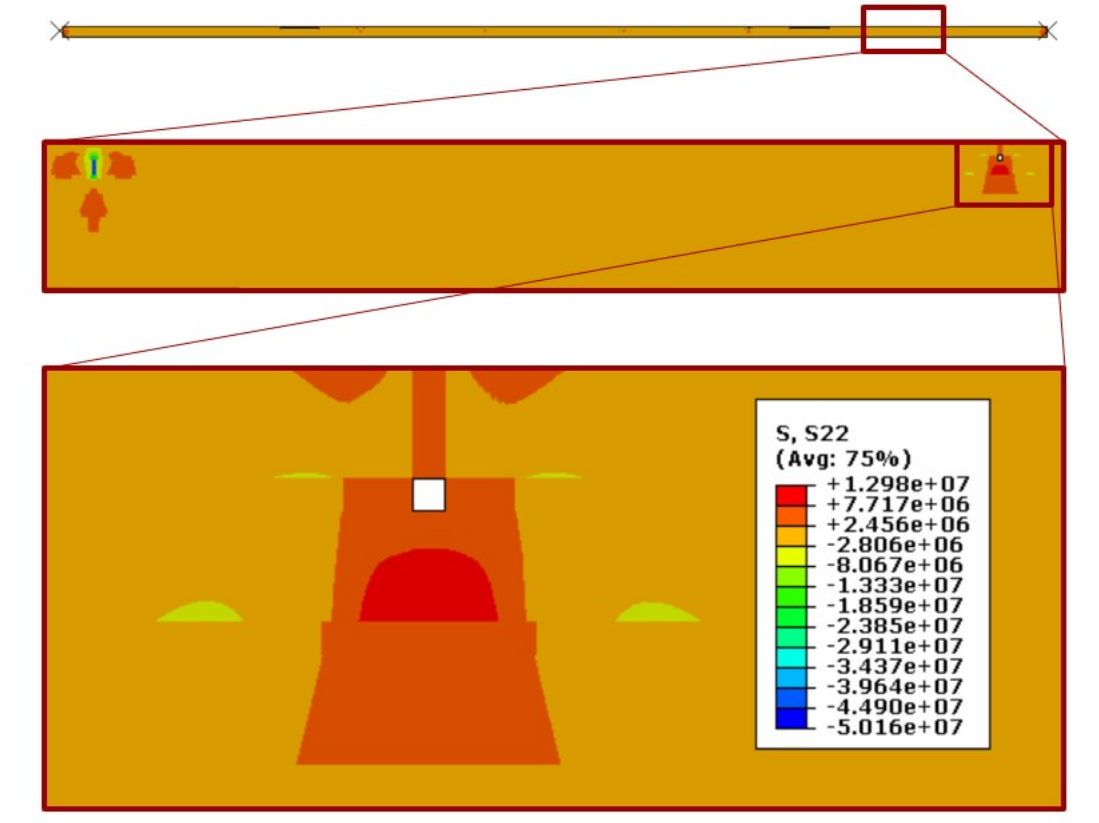


Figure 33: Peel stress

The shear stress S12 is shown in Figure 34

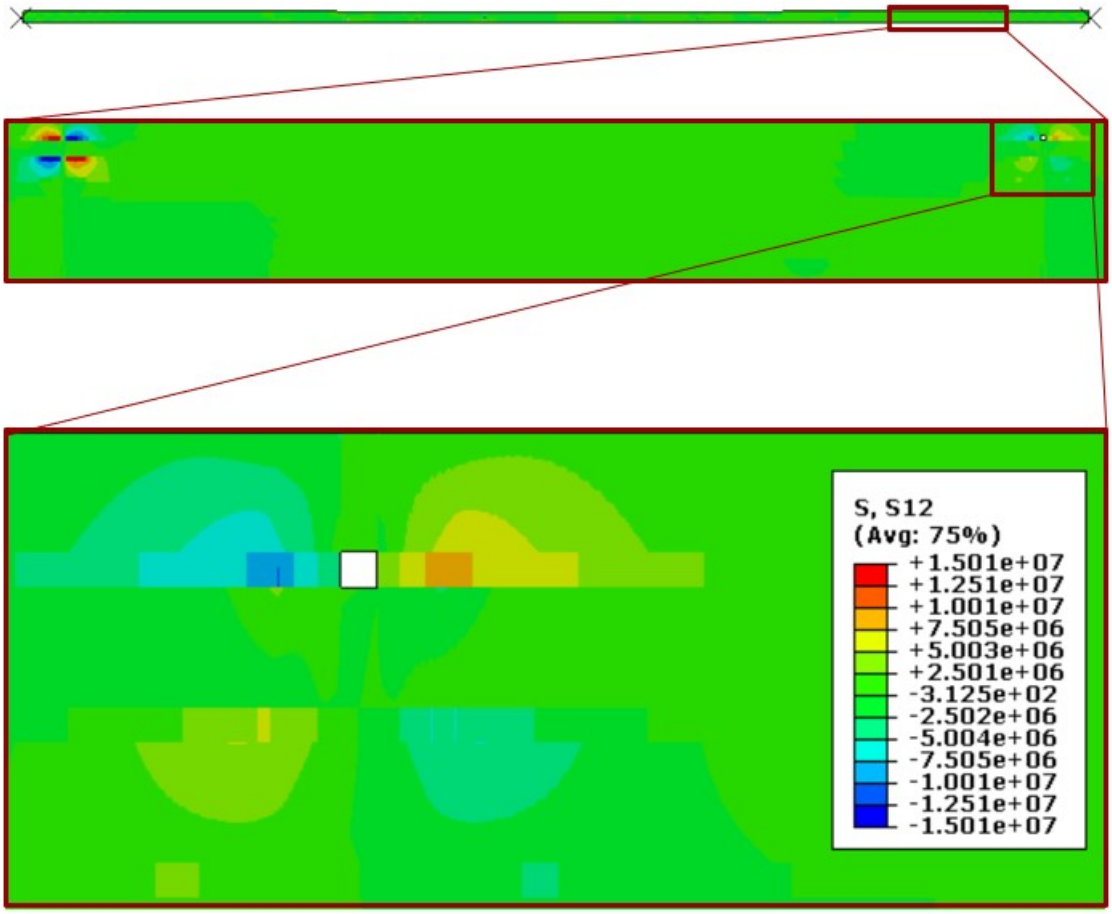


Figure 34: Shear stress

This observation is in agreement with the experiments done by [4] and with [8]. No debonding was observed in the BIAX plies.

- **IV**: The fracture energy limit is reached for the first time in an element close to the second vertical resin element. The crack starts to grow to the left, in the direction of the 3rd step, and to the right where it will meet the crack of the 1st resin layer. The crack of the 1st resin layer continues to grow. The failure of the first element close to the second vertical resin element is shown in Figure 35

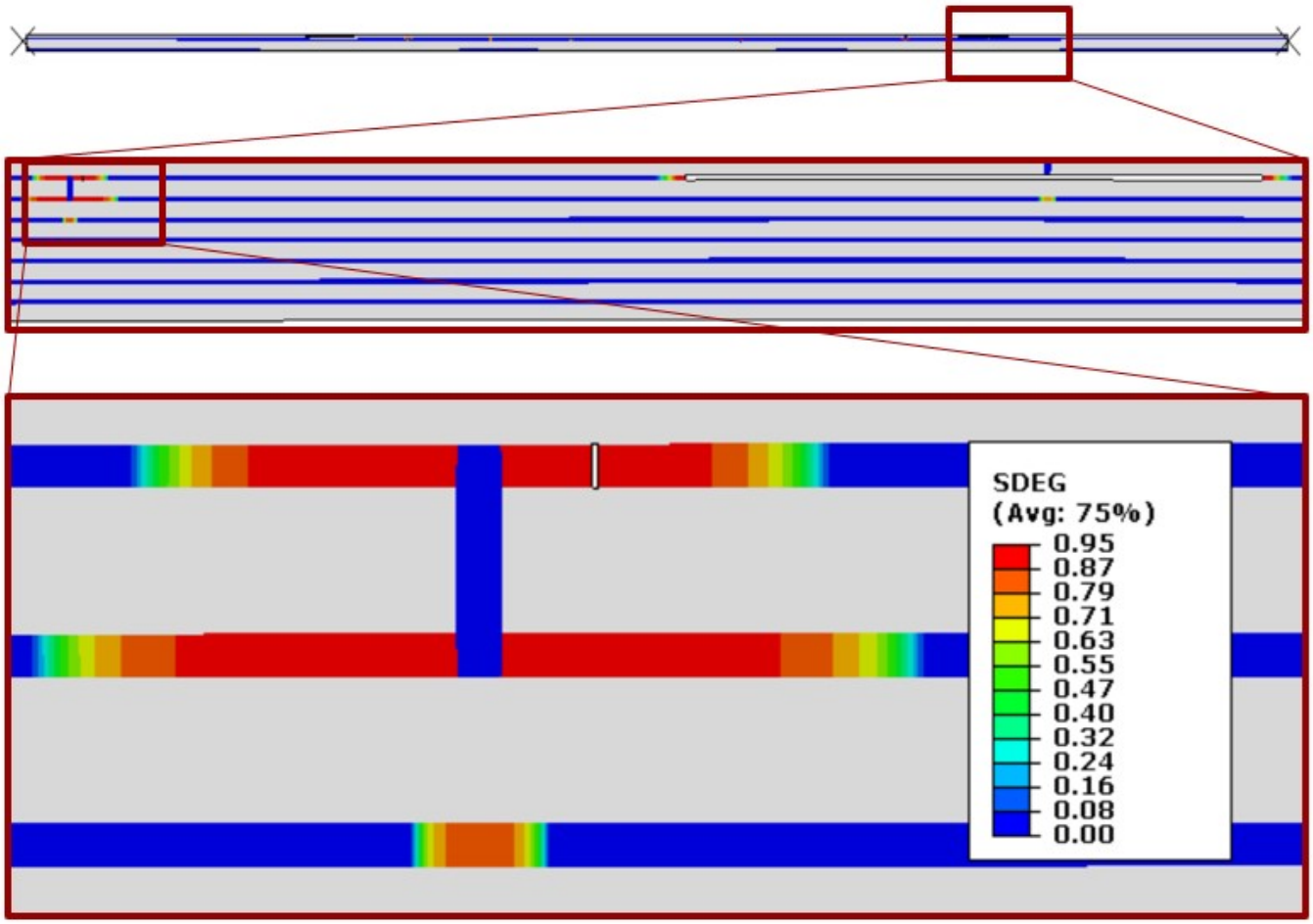


Figure 35: Crack appearance in the second step

- **V**: The crack of the 1st vertical resin element meets the crack of the 2nd vertical resin element. At this moment, the load carried by the UD fibers of the repair ply and the UD fibers of the parent ply loose their contact surface, and as a result, the laminate suffers a sudden drop in strength. During this stage the maximum load is reached at 5.0629e+05 N. In Figure 36 the point at which the two cracks meet is shown.

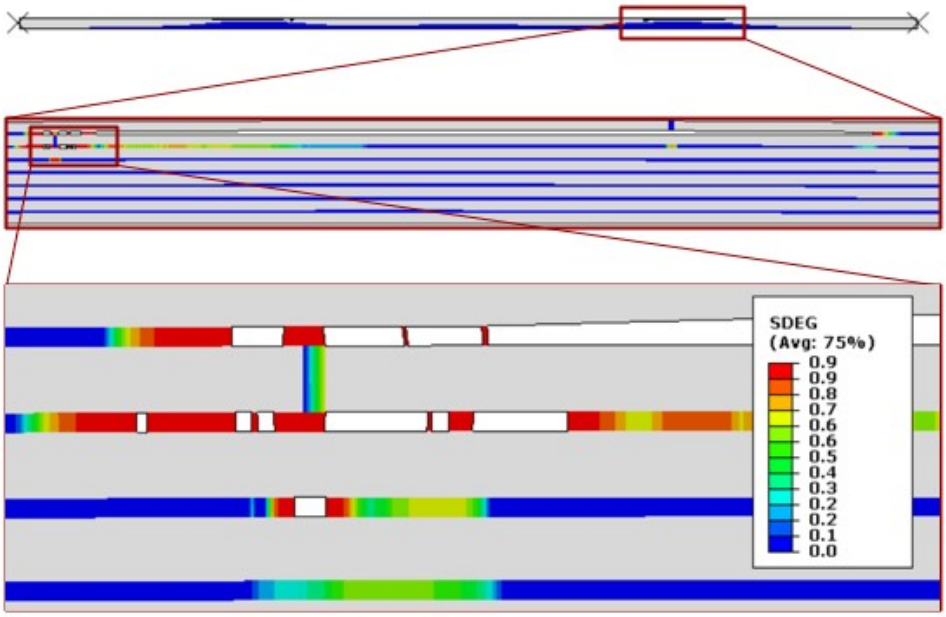


Figure 36: Crack meeting point

During this short stage the stiffness of the laminate changes and a non linearity stage occurs as shown in Figure 37:

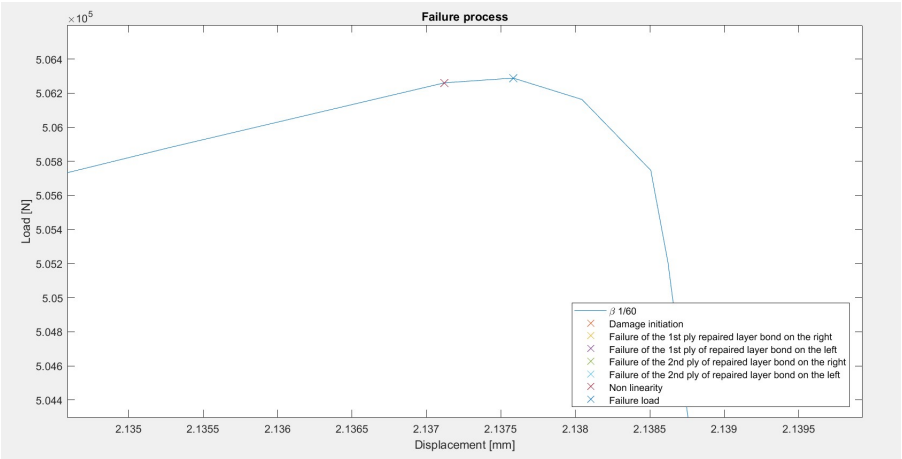


Figure 37: Non linearity

- **VI**: During this stage the laminate looses all its strength and the load drops. This caused by the debonding of the plies, more precisely, by the detach of the contact surface betweeen the UD repaired ply and the UD parent ply. As a consecuence, the other plies have to suddendly carry more load. This causes the second resin ply from the top to also debond as shown in Figure 38

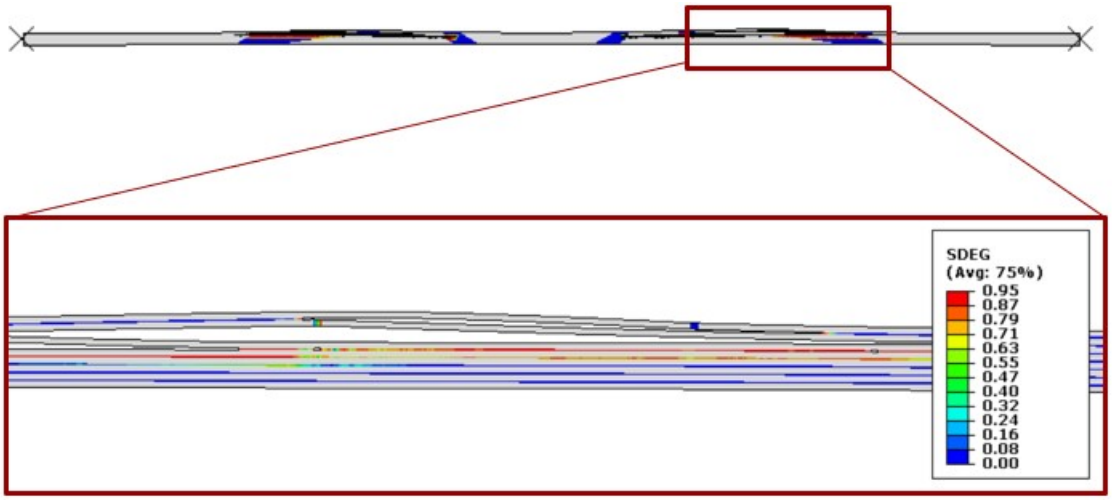


Figure 38: Total failure of the laminate

The delamination process is showed in the video Figure 39. Click on the image to go to watch the video. If the image does not work, the link is: `https://www.youtube.com/watch?v=pzpZ9xWwQDs`

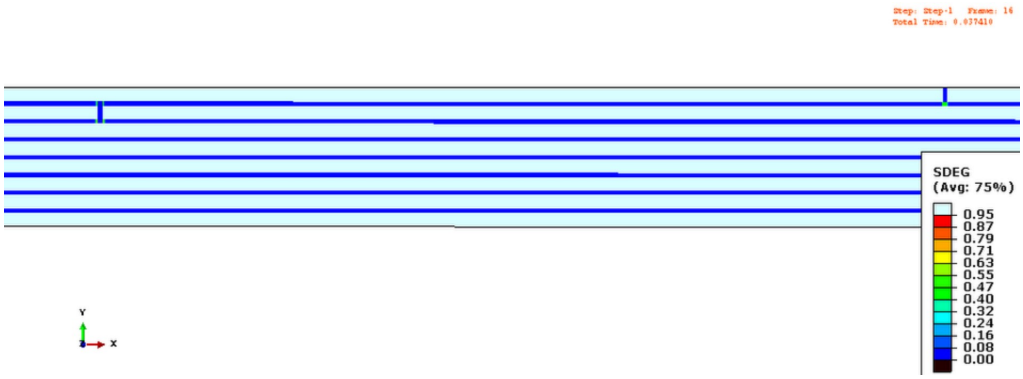


Figure 39: Video of the delamination

5.2.4 Mesh convergence

The FE program results are sensitive to the mesh size. If the mesh size is coarse, it can lead to errors in the simulation. In this section, the mesh convergence of the case $\beta = 1/60$ is checked. The following Figure 40 shows the convergence. Since the mesh has different sizes depending on the distance to the vertical resin element, the smallest element of the mesh was taken as a reference.

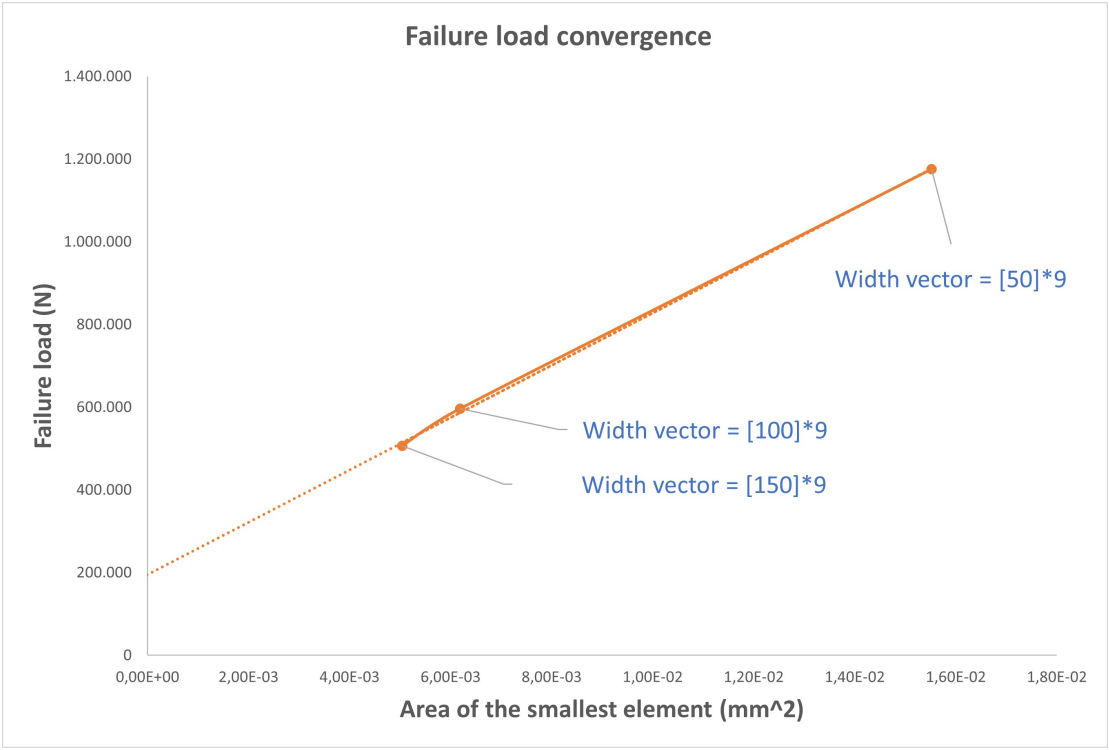


Figure 40: Mesh size convergence

The bigger the width vector values (Appendix A Figure 3.2), the smaller the element size is. From Figure 40, it can be observed that the highest width vector case for a value of 150 failed before the case for a width vector of 100. The width vector case of 100 failed before the width vector case of 50. These results match what is expected to happen since the smaller the mesh size is, the smaller the error is, and the closer the simulation is to the natural physical phenomena. It is important to remember that results FE overestimate the potential energy. The total potential energy of an approximate solution is always larger than that of the true solution:

$$\Pi(\{D\}) \geq \Pi_\infty \text{ for } \forall\{D\} \neq \{D_\infty\} \tag{3}$$

$$\{D\}^T\{P\} \leq \{D_\infty\}^T\{P\} \text{ for } \forall\{D\} \neq \{D_\infty\} \tag{4}$$

The approximate FE solution is always too stiff, but the error gets smaller, the smaller the mesh size is [16].

5.2.5 Results validation

As discussed in [4] β is a important variable because it affects the area of the contact surface between the parent and the repaired plies, therefore, the transmission of the load along the repaired laminate. The chosen values for β were: 1/60, which corresponds to the value used by Airbus [15], 1/45, which corresponds with Boeing [15] practices, and 1/30, which was chosen to study the effect of a larger β on the laminate strength.

Experimental data suggests that larger β values result in less strength efficiency [4]. The same conclusion was achieved with the ABAQUS simulations as shown in Figure 41. The smallest β value of 1/60 achieved a higher load before failure, then the next one to fail was the laminate with the β ratio of 1/45, and finally, the first one to fail was the laminate with the β ratio of 1/30. This supports the validation of the laminate model that uses a cohesive element type as the resin layer bond.

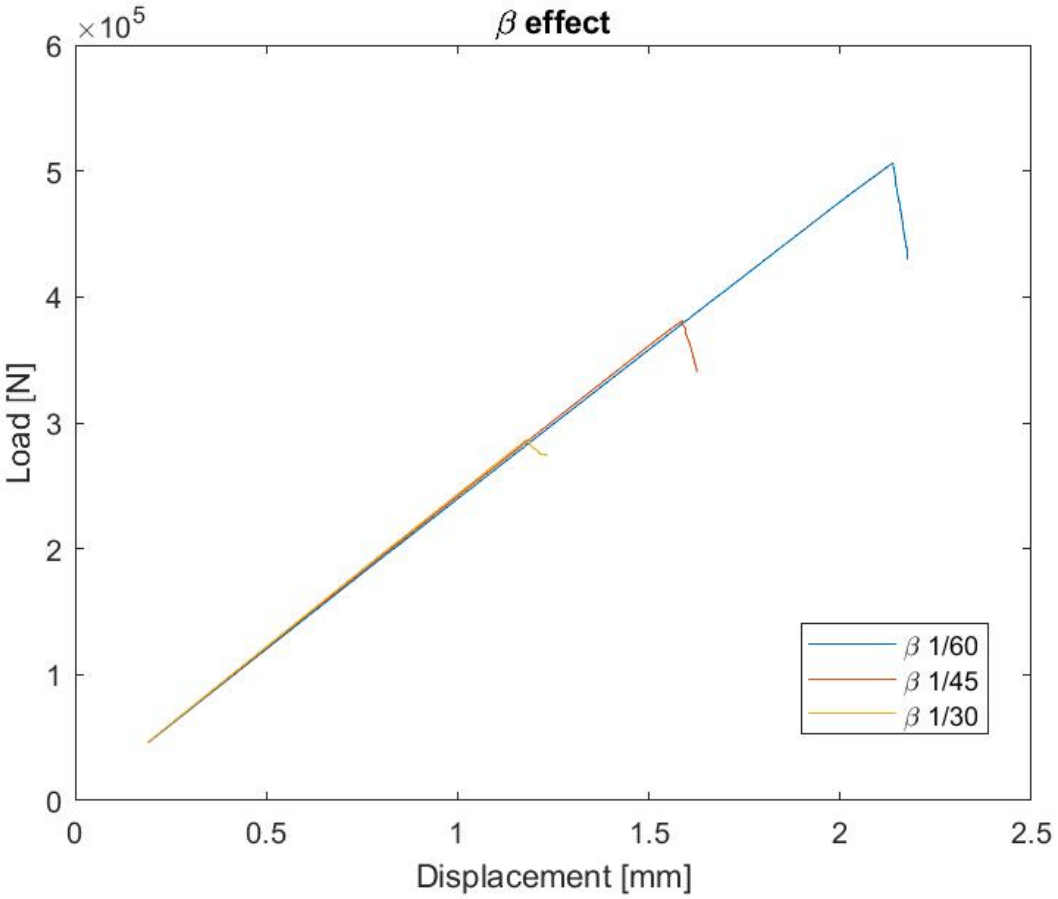


Figure 41: β effect to the strength of the laminate

6 Conclusions and Future work

6.1 Conclusions

The MPSS program has been created. It allows users to simulate a step scarf repaired laminate with the desired parameters. The results have been validated with other scientific papers that included both FE modeling and experimentation.

The results provided by the FE simulation for a β of 1/60 support the results obtained in [4] and [8], and are an example of how important is the overlapping layer length to the final strength of the laminate. The larger the overlapping layer is, the larger the crack would need to be to separate and detach the contact surface between the plies carrying the load. In this case, the debonding that caused the failure was between the UD repair and parent plies. The final objective of the repair is to have the stiffness and the strength restored. However, the stiffness it better restored with a shorter overlapping layer length [13] and the strength is better restored with a longer overlapping layer length, so a mid term solution between a long and short overlapping layer length appears to be the best option.

6.2 Future work

The MPSS program is complete, but there are some improvements that can be made to it.

- **3D simulations**: right now, the MPSS creates input files in 2D; it would be a good upgrade if it was developed for 3D also in the future. It would be helpful for 3D wrinkled plies where not the same area of the defect is removed in each plane.
- **More number of plies**: right now, the MPSS can only use UD and BIAX at the same time, even though they can be different between the parent UD and the repair UD plies. In reality, 90°plies are commonly used with 0°plies and 45°plies, and to simulate most cases, a third type of material is needed.
- **Mismatch**: right now, the mismatch option works by flipping the element sets between UD and BIAX. However, there are cases in which a laminate contains consecutive plies of the same type, i.e., [BIAX, BIAX, UD, UD]. In this case, flipping the element sets does not work.
- **Optimization**: since the MPSS is a program that creates, sends the input file to ABAQUS, and then reads the output, it could be possible to loop the process to obtain the most desirable laminate geometry. The user would be able to input a desirable strength for the laminate after the repair, and the program would loop simulations to get the optimum geometry. If this improvement was made, the program would take a leap to another level in terms of usefulness.

7 GUI documentation

7.1 General Input

7.1.1 Input file name

In this section, the user is asked to write the name of an excel file and a number for the sheet. These data should correspond to the data needed for the wrinkle representation. This sheet should look like CASE1.xlsx sheet 1. The first column represents the x direction and the second column represents the height of the wrinkle that is used as a defect.

7.1.2 Output file name

In this section, the user is asked to write the name of the files that the MPSS will create. This name will be the one that the: .inp, .odb, .mdl, .sim, .prt, .msg, .com, and . dat files will have.

7.1.3 Ply by ply match

In this section, the user can choose whether the repaired laminate plies match. The mismatch is done vertically, as in figure Figure 42.

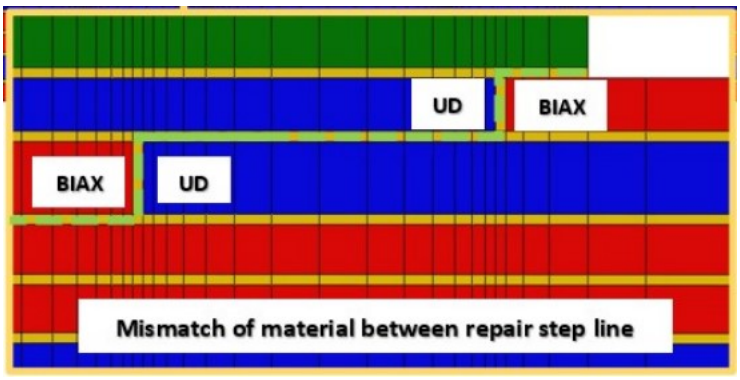


Figure 42: Mismatch representation [13]

It is an axiom of this type of repair that the material stiffness of the overlapping repair ply should match that of the underlying parent ply so that load transfers successfully through the stack of plies laminate.[4]

7.1.4 Width defect

In this section, the user can define the length of the defect in the X direction. It will affect the scarfing area.

7.1.5 Overlamina addition on top of the repair

In this section, the user can choose to have an extra layer on top of the already existing ones. The layer can be with or without an overlapping layer length. If the user chooses not to apply the overlapping layer length to the overlamina, the previously defined overlapping layer length of the overlamina will not be used.

7.1.6 Curve panel

In this section, the user can choose to have a curve laminate instead of a flat laminate. The laminate curves using the following equation:

$$y = \frac{\sqrt{xcurveL}}{\sqrt{a_curve}}$$

Figure 43: Curvature formula

7.1.7 Resin element

The user can choose between solid and cohesive elements for the resin layers in this section. The cohesive should be more accurate since the resin layer elements will be COH2D4 (cohesive zone elements) instead of CPE4 (Four-node plane strain element). In addition, the failure of the laminate can be studied with the cohesive elements.

7.1.8 Loading

In this section, the user can choose which type of force will be applied to the laminate in ABAQUS.

- Loading = 0 The repaired composite will be subject to an axial force caused by a displacement in the X direction. The displacement is applied at the right end of the laminate. The left end of the laminate is fixed with $\Delta x = 0$
- Loading = 1 The repaired composite will be subject to a rotation applied in the pinpoints. The boundary conditions are the same as a simply supported beam. The rotation direction is clockwise for the left end and counterclockwise for the right end. In case cohesive elements are chosen by the user, ABAQUS may take a longer time to run.

7.1.9 Gaussian parameters guess

The user can define the parameters needed for interpolating the wrinkle in this section. The wrinkle is defined by the input excel file with some points. The gaussian approximation creates a curve out of those points. The parameters are just a guess for the gaussian approximation. However, they will speed up the algorithm since the code will find the correct curve before that if the parameters are wrong.

7.1.10 Number of CPU cores used for the simulation

In this box, the user can change the number of CPU cores that the computer will use to run the simulation. The more cores used, the faster the simulation will be.

7.2 Geometry

7.2.1 Layup selection process

In this section, the user can choose to create four different types of .inp files depending on the flag option. The flag meaning is the following:

- Flag = 0 Composite with defect and layup contains two different lamina materials.
- Flag = 1 Composite with defect and layup contains one type of lamina material.
- Flag = 2 Composite without defect and layup contains two different lamina materials.
- Flag = 3 Composite without defect and layup contains one type of lamina material.

The flag type must match with the next input, which is layup with defect and layup without defect, e.g., if $Flag = 0$, then UD and BIAX should be included in the layup with the defect.

The input for the layup with and without defect must be in lowercase letters and without a space between the ',' e.g., biax,ud,biax. The plies will be created from top to bottom.

There is also an option to change the repair material in the parent material drop-down. It is useful in case the new material is not the same as the parent plies. e.g., the repaired composite laminate will not have the old material for the repaired layers because the old material is not available.

7.2.2 Overlapping layer length

The user is asked to write the overlapping length for the different materials in this section. This overlapping length represents the contact surface between the old laminate and the newly added plies.

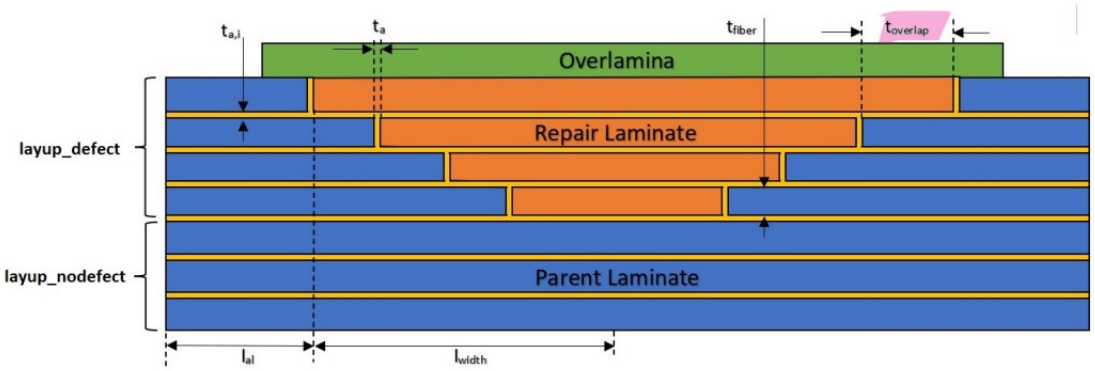


Figure 44: Overlapping layer length [13]

7.2.3 Define material thickness

In this section, the user is asked to provide the thicknesses of the different plies. The resin interface between layers represents, as its name suggests, the resin located between the different plies stacked vertically in the Y direction (tHres). The thickness of the UD (tud) and BIAX (tbiax) materials should be different for the code to work. If the user wants them to be the same thickness, the code will work for a small difference, e.g., UD thickness = 0.8 and BIAX thickness = 0.7999. The resin between the repair and parent laminate represents the resin layer between the old and the newly added plies (tVres). The thickness of the overlamina should be 'ud' or 'biax'. If the overlamina layer is not added, this value will not be used.

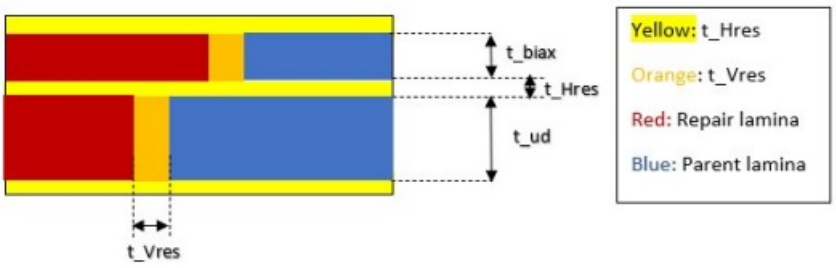


Figure 45: Different thicknesses [13]

7.2.4 Geometry of the 2D model

In this section, the user can define the size of the laminate from the middle where the defect is defined, e.g., If the width from the middle of the defect is 500 mm, then the size of the laminate will be 1m. The user can also define an additional allowance, which will be added on the left side and the right side of the laminate.

7.2.5 Element number

In this section, the user can define how is the mesh distributed. The laminate is divided into different zones. This division is given by the resin layer of each ply that separates the old plies from the new plies. For example, if the layup with defect is biax,ud,biax, then there will be: $N°zones = 3 * 2 + 1 = 7$ The general equation is the following:

$$N°zones = N°plies * 2 + 1 \tag{5}$$

The width vector number contains information of how many elements there will be for each section.

7.2.6 Discretasing density

In this section, the user can change the density that the ABAQUS mesh will have. This number must be bigger than the sum of all the elements in 'Element number', e.g., if element number is '80,80,80' then the discretasing density must be bigger than 240.

7.3 Material properties

In this section, the user can introduce the UD, BIAX, and RESIN material properties.

The extra UD and BIAX properties from cohesive material properties are only used if the user has selected parent material = 1 with different material in General input. These material properties will be used for the repaired plies instead of the properties from Material properties. In case the user has chosen the parent material = 0 with the same material, then both repair and parent plies will use the material properties from material properties.

7.4 Cohesive material properties, General Static Analysis and Step control

7.4.1 General Static Analysis

The general static analysis parameters are ABAQUS variables that affect the convergence of the simulation. If the step minimum time increment is big, or the maximum time increment is also too big, then the simulation might not converge. The meaning of each variable is the following

- **1st step**: this is the increment size that the first step will have. If time period is for example, 1s, then if 1st step is 0.1s, the 1st step will have load of 10% of the total load.
- **Minimum time increment**: this is the smallest step that the simulation is allow to take. For example, if time period is 1s, and minimum time increment is 1⁻10, then the minimum increment load is allow to take is 1⁻10 of the total load.
- **Maximum time increment**: this is the biggest load increment that the simulation is allow to take. For example, if time period is 1s, and maximum time increment is

0.1s, then the biggest load increment that the simulation will take for a single step is a 10% of the total load.

- **Time increment**: it is the length of the load curve that ABAQUS will apply to the laminate.

7.4.2 Cohesive elements behavior

If cohesive element option is chosen, the damage evolution criterion that will be used is Benzeggagh-kane. The user inputs η. The following figure explains the behavior.Figure 46:

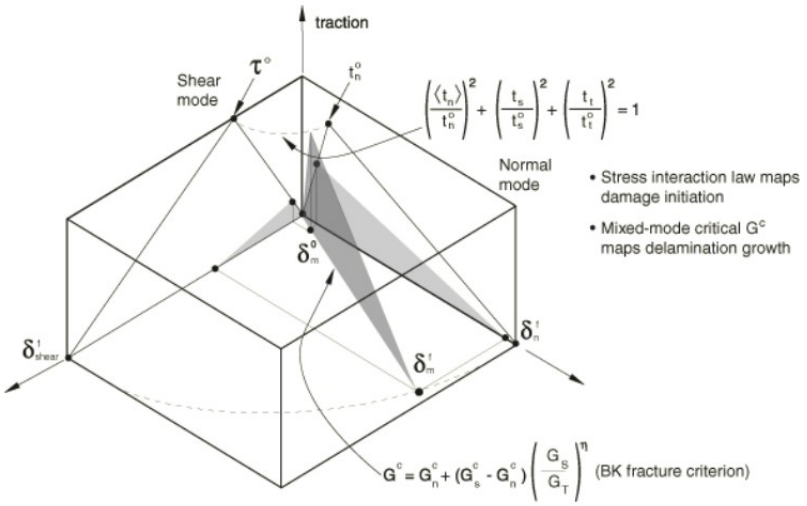


Figure 46: Mix mode response in cohesive elements [17]

7.4.3 Damage initiation criterion: QUADS & MAXS

The resin elements are evaluated using QUADS and the plies are evaluated using MAXS.

Quadratic nominal stress criterion (QUADS): the user introduces the maximum stress values. When the Equation 6 reaches 1 the damage initiation limit is reached and the damage starts.

$$\left\{\frac{\langle t_n \rangle}{t_n^o}\right\}^2 + \left\{\frac{t_s}{t_s^o}\right\}^2 + \left\{\frac{t_t}{t_t^o}\right\}^2 = 1 \tag{6}$$

Maximum nominal stress criterion (MAXS): the user introduces the maximum stress. Damage is assumed to initiate when the maximum nominal stress ratio is 1 . This criterion is represented in the following Equation 7:

$$\max\left\{\frac{\langle t_n \rangle}{t_n^o}, \frac{t_s}{t_s^o}, \frac{t_t}{t_t^o}\right\} = 1 \tag{7}$$

7.4.4 Damage evolution: Fracture energy.

In this box, the user can introduce the values for the fracture energy G_n, G_s, and, G_T for UD, BIAX, and resin elements. Damage evolution can be defined based on the energy that is dissipated as a result of the damage process, also called fracture energy. The fracture energy is equal to the area under the traction-separation curve Figure 47. You specify the fracture energy as a material property and the behavior is linear. ABAQUS ensures that the area under the linear damaged response is equal to the fracture energy.

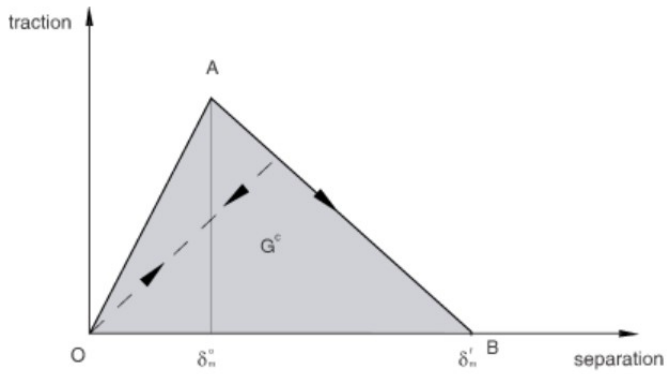


Figure 47: Fracture energy [17]

The maximum energy on the mode mix can be specified directly in window Cohesive material properties. When the analytical forms are used, the mode-mix ratio is assumed to be defined in terms of energies. The damage evolution behavior is as follows ([17]):

$$\begin{aligned} t_n &= \begin{cases} (1-D)\bar{t}_n, \bar{t}_n \geq 0 \\ \bar{t}_n, \end{cases} \quad \text{otherwise (no damage to compressive stiffness)} \\ t_s &= (1-D)\bar{t}_s \\ t_t &= (1-D)\bar{t}_t \end{aligned} \tag{8}$$

Where t_n, t_s, and t_t are the stress components predicted by the elastic traction-separation behavior for the current strains without damage [17].

7.4.5 Linear elastic traction-separation behavior

In this box, the user can introduce the values for the elastic behavior of the resin element.[17]. They represent the slope of the elastic portion of the traction separation law. The option is used to define uncoupled traction-separation behavior. It is only available when the cohesive elements option is chosen in the General input window.

$$\begin{Bmatrix} t_n \\ t_s \\ t_t \end{Bmatrix} = \begin{bmatrix} E_{nn} & & \\ & E_{ss} & \\ & & E_{tt} \end{bmatrix} \begin{Bmatrix} \varepsilon_n \\ \varepsilon_s \\ \varepsilon_t \end{Bmatrix} \tag{9}$$

The quantities t_n, t_s, and t_t represent the nominal tractions in the normal and the two local shear directions, respectively; while the quantities ϵ_n, ϵ_s, and ϵ_t represent the corresponding nominal strains.

7.4.6 Damage

This box includes the damage stabilization for the XFEM simulation in case it is done in the future (without modeling a crack in the fibers the resin layer will always fail before the fibers). It also includes the maximum stiffness degradation or SDEG at which ABAQUS will delete the cohesive elements

7.4.7 Step control equilibrium: Time incrementation

Includes I_o and I_R which help with the step convergence. I_o is the iteration for a residual check and I_R is the equilibrium iteration for a logarithmic rate of convergence check. [17]

7.5 Run/Output

7.5.1 MENUa and output

In this section, the user can run the GUI and restart it by clicking on their respective buttons. The JOB label has three states: Running ABAQUS, DONE, and Error. Running ABAQUS will be written when the code sends the .inp file to ABAQUS to execute it. DONE will be written when the MATLAB and ABAQUS are done, the results are read, and the equivalent stiffness of the laminate or the bending stiffness are calculated again by MATLAB and shown in the GUI. Figure 48

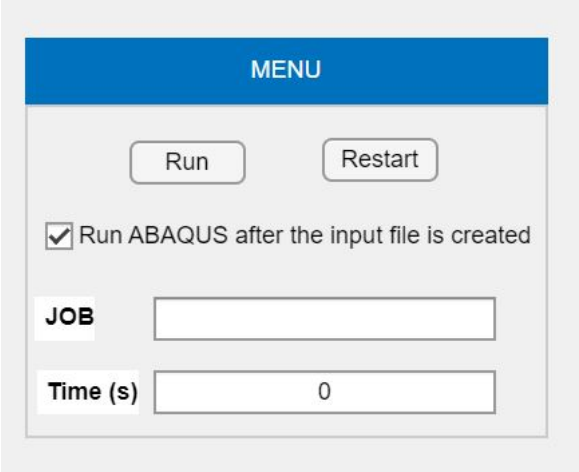


Figure 48: GUI MENU

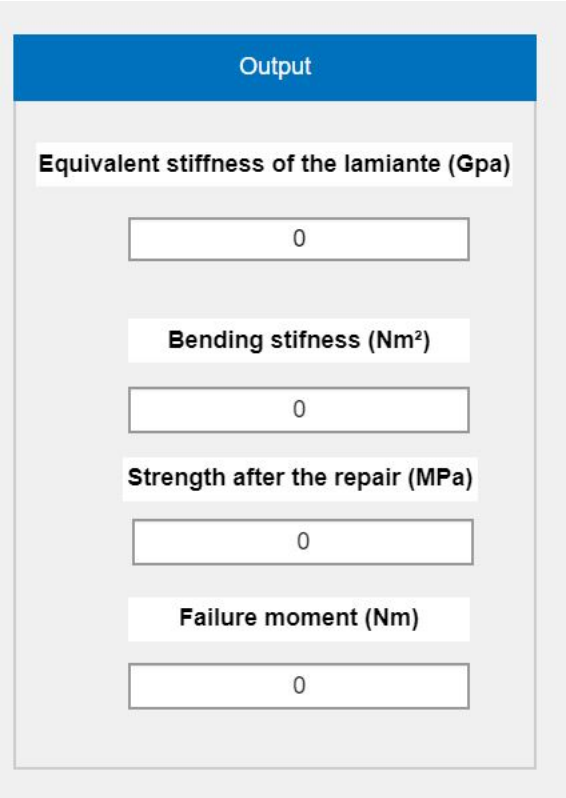


Figure 49: Output

Note: JOB will show Error if there is an error in the MPSS part (MATLAB). If the .inp file is incorrect, the JOB state will be Running ABAQUS because MATLAB cannot know if ABAQUS has an error. In case running the ABAQUS state takes a long time, go to the command window in MATLAB to see if the ABAQUS gave back an error. If a cohesive analysis is done, the ABAQUS will always give back an error when the laminate fails because it is unable to calculate the next step increment. In case the state is Error, the flag number and the layup with or without defect probably do not match, or the discretasing density is smaller than the sum of element number vector.

7.5.2 Plot

The GUI will show the first plot while the code is still running to show that the gaussian approximation is made. However, the user will not be able to change to the other plots in Choose plot drop-down (Figure 50) until the code has finished running. If the user wants to see the Load vs. Displacement curve, the engineer should select the 8^{th} option in the Choose plot drop-down. The other plots are the steps the MPSS takes to create the repaired laminate.

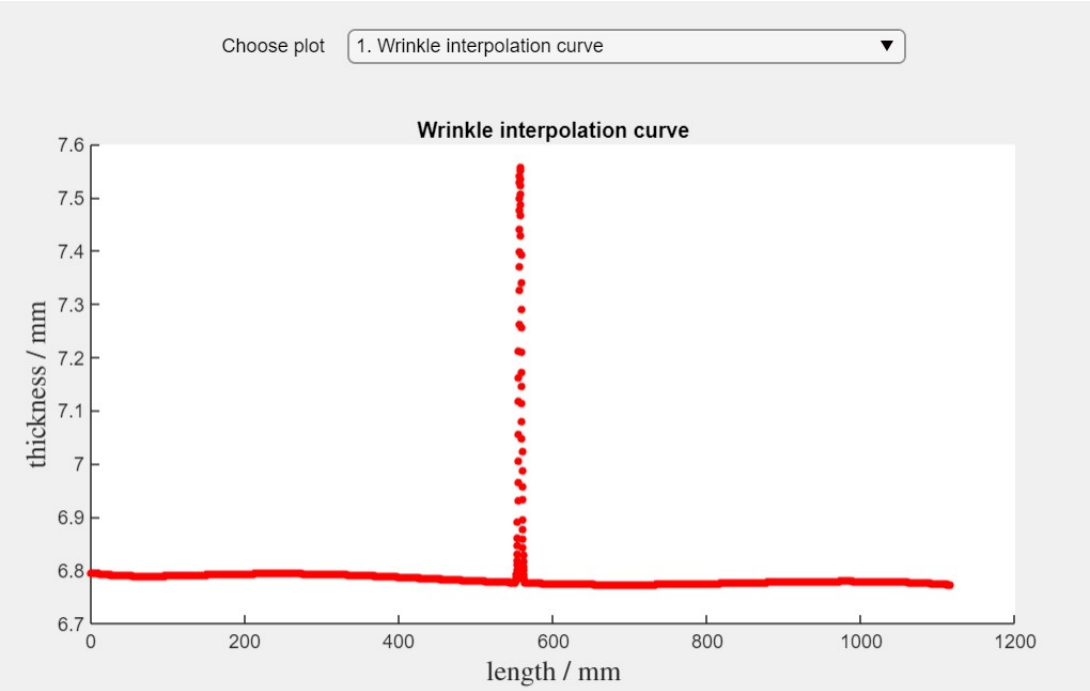


Figure 50: MPSS Plot

Note: If the user wants to see a plot that has already been plotted, the plot graph may have the same zoom as the plot from before. If that is the case, the user can solve the problem by zooming in or out.

7.6 Output: Stress and Strain

In this section Figure 56, the user can see the different stresses and strains of the different element sets that ABAQUS returns after the simulation. The reaction forces can also be seen on the right side.
The different element sets are the following:

- 1. EINTERFACE: element set that refers to the resin between the plies (tHres).
- 2. EOVERLAM: element set that refers to the extra ply added at the top of the lamina.
- 3. EPARENTBIAX: element set that refers to the parent BIAX plies.
- 4. EPARENTUD: element set that refers to the parent UD plies.
- 5. EREPAIRBIAX1: element set that refers to the replaced BIAX plies when the mismatch option is selected.
- 6. EREPAIRUD1: element set that refers to the replaced UD plies when the mismatch option is selected.
- 7. ERESIN: element set that refers to the resin between the old plies and the newly replaced plies (tVres).
- 8. RESINRICH: element set that refers to the resin pocket created when the mismatch option is selected.
- 9. EREPAIRBIAX: element set that refers to the repaired BIAX plies when the mismatch option is not selected.
- 10. EREPAIRUD: element set that refers to the repaired UD plies when the mismatch option is not selected.
- 11. ECOMPOSITEBIAX: element set that refers to the BIAX material when the pristine model is selected.
- 12. ECOMPOSITEUD: element set that refers to the UD material when the pristine model is selected.

7.7 GUI interface

7.8 Appdesigner

The graphical user interface (GUI) is developed in MATLAB Appdesigner. This software environment allows the developer a more straightforward method to develop the GUI than the old GUIDE environment. To open the environment, type *appdesigner* in the command window in MATLAB.

The Appdesigner environment fills for the app developer the necessary code for the different boxes when dragged into the interface; as a result, that code is fixed and unchangeable by the developer or user. All the code is shown in Figure 64

7.9 GUI sections

In this section, the different windows of the GUI are shown.

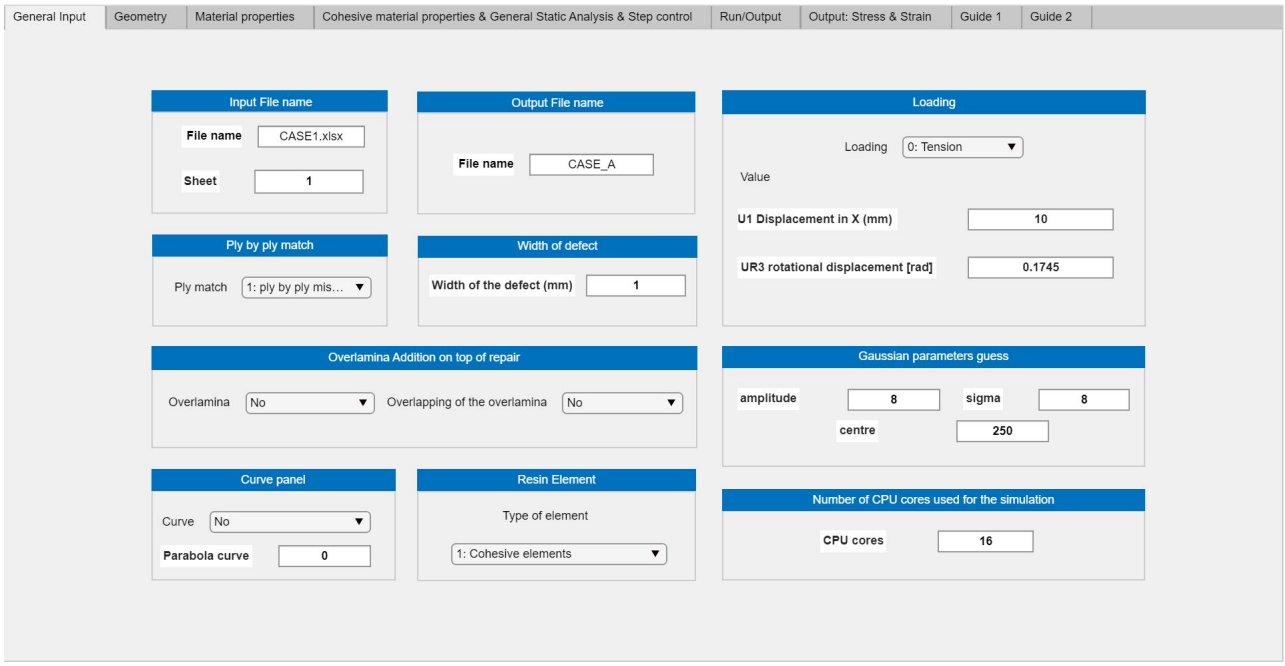


Figure 51: General input

In window Figure 51, the user inputs the general data and specifications of the desired laminate.

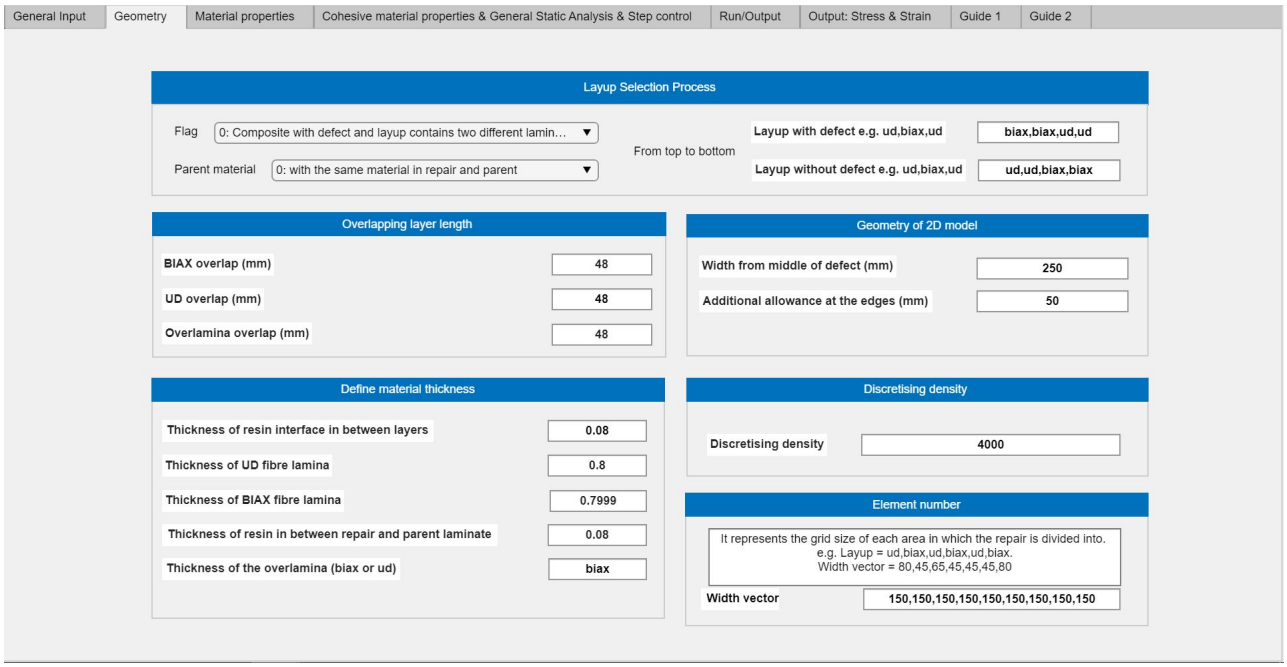


Figure 52: Geometry

In window Figure 52, the user inputs the geometry of the desired laminate.

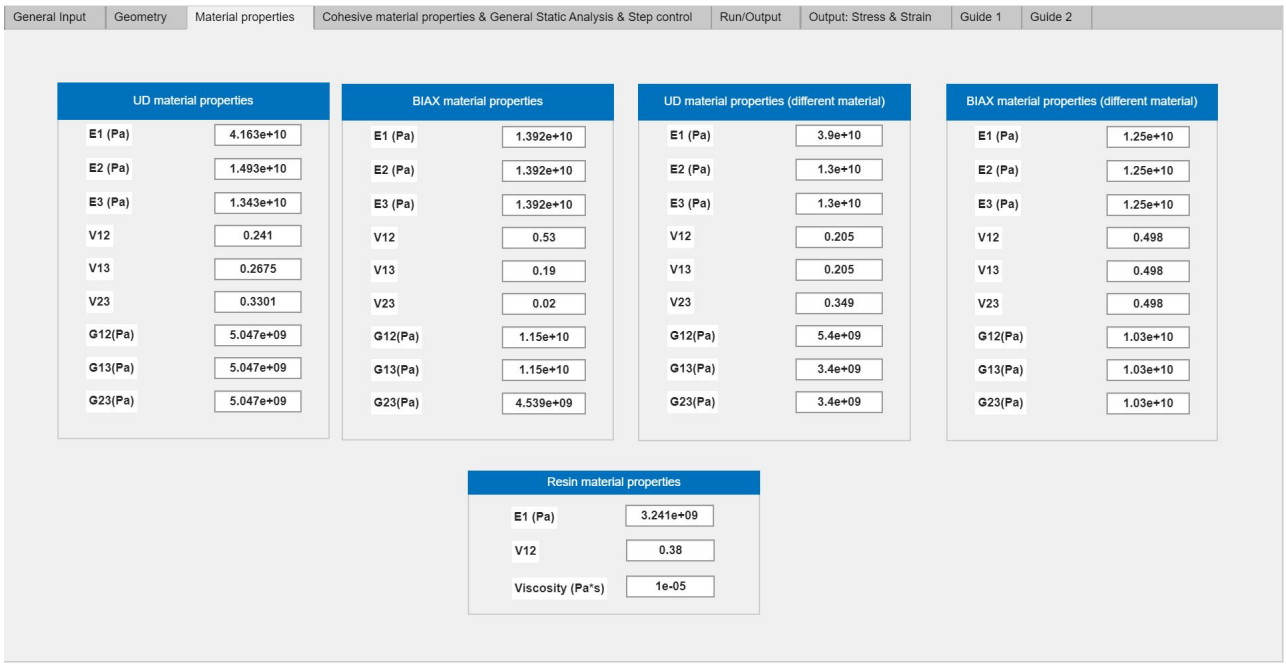


Figure 53: Material properties

In window Figure 53, the user inputs the material properties of the desired laminate for UD, BIAX, and resin materials.

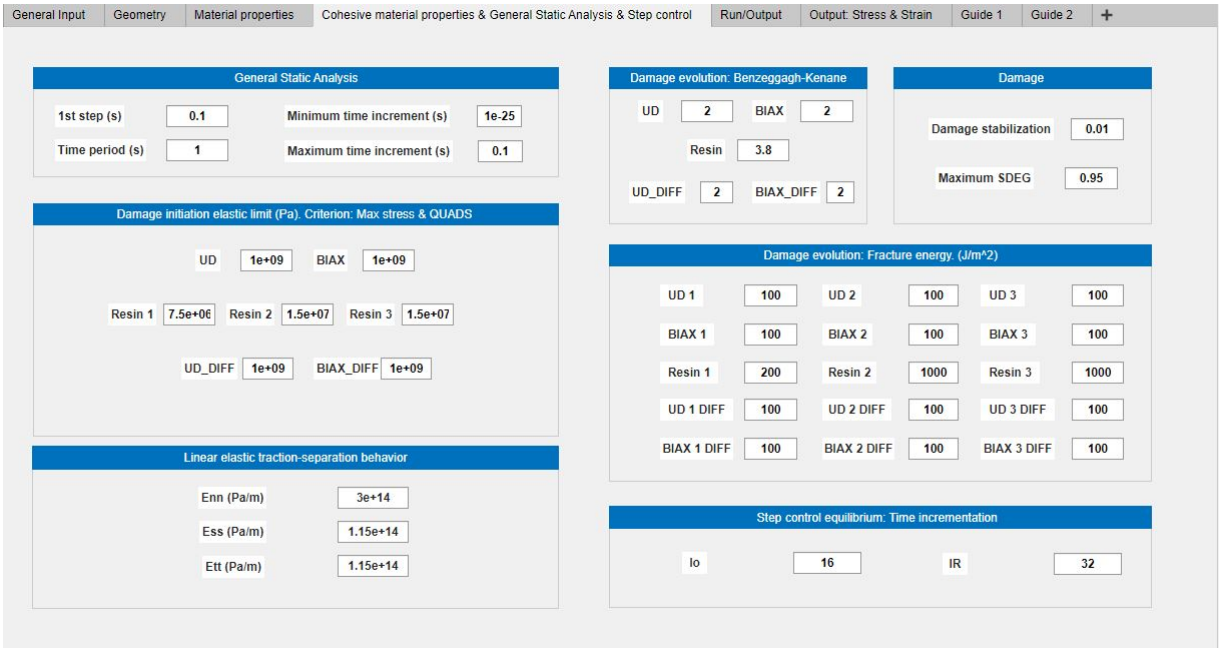


Figure 54: Cohesive material properties

In window Figure 54, the user input the cohesive material properties of the desired laminate for UD, BIAX, and resin materials.

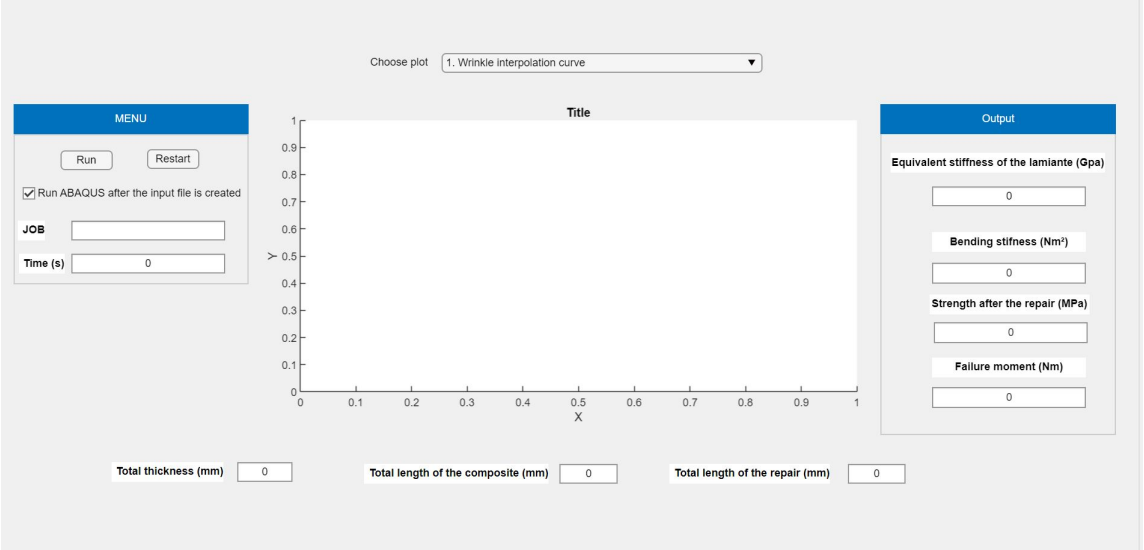


Figure 55: Run and Output

In this window Figure 55, the user can use the menu and see the MPSS algorithm's output. The graphs show the steps to replace wrinkled plies with healthy ones. Using the drop-down, the user can change between the different steps and see their graphs. The last option of the

Choose plot drop-down includes a Load vs. Displacement graph. On the bottom, the user can see the final geometrical parameters of the laminate.

Figure 56: Output: Stress and Strain

In this window Figure 56 the user can choose in the drop-down between the different element sets of the lamina. This window can be used without running the code again if the .dat output file already exists. The code will look for the maximum absolute value for each of the stresses and strains. In case an element set that is not present in the .dat file is selected the 'JOB' edit field will give the appropriate error. The .dat file will only contain the element sets that have been checked in the box.

Figure 57, Figure 58 and Figure 59 are a guide for the user to understand the variables correctly and to learn how to use the app.

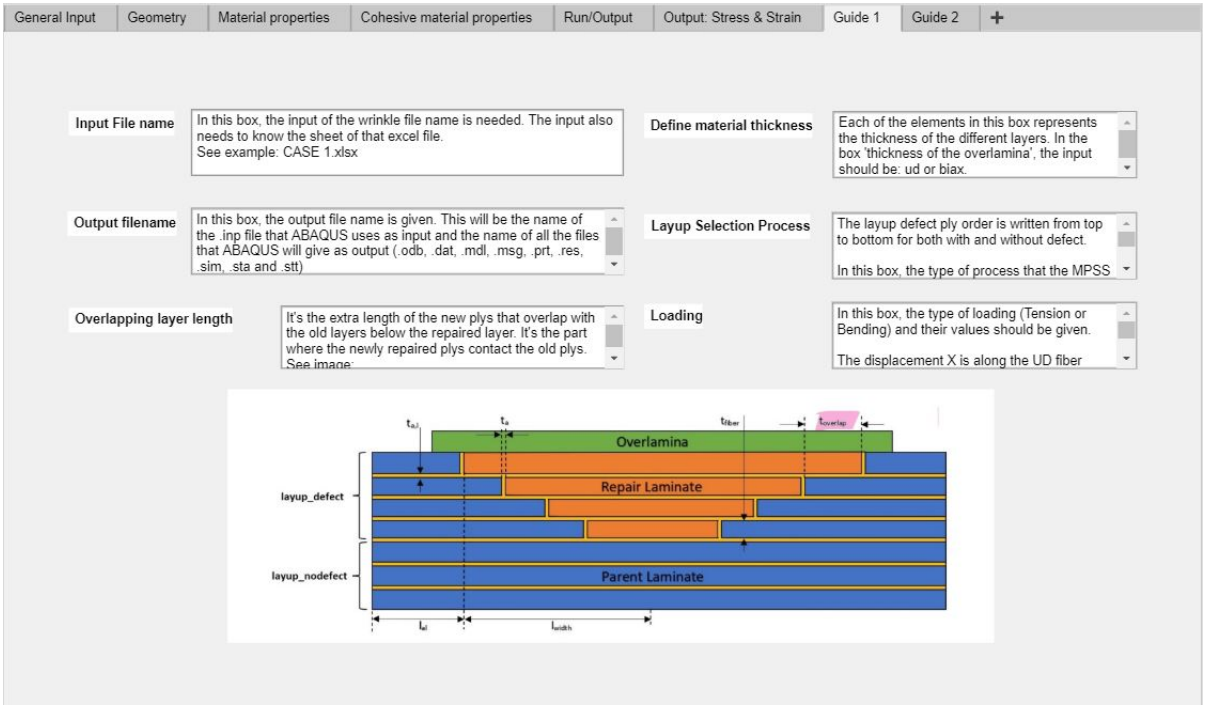


Figure 57: Guide 1

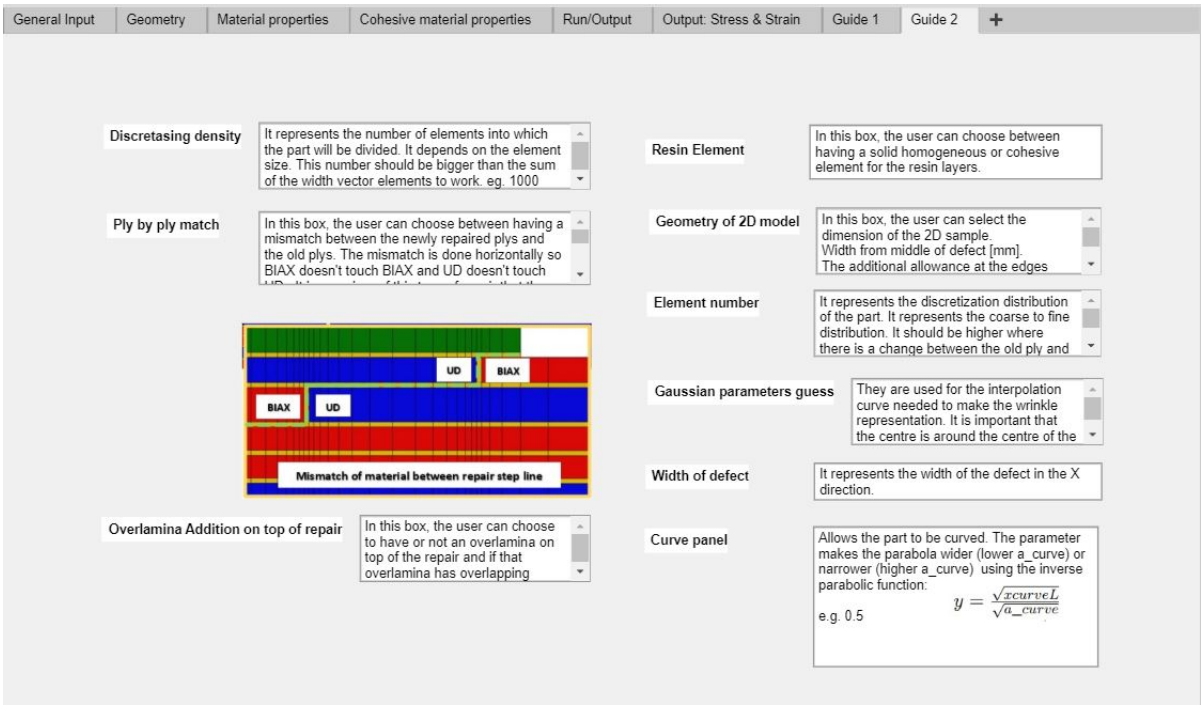


Figure 58: Guide 2

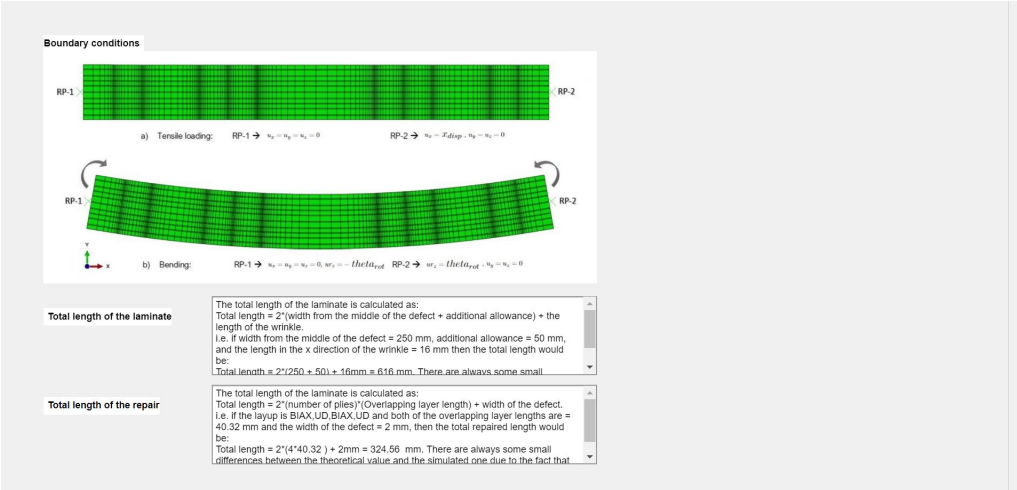


Figure 59: Guide 3

7.10 Loading and boundary conditions

There are two types of boundary conditions depending on the load: tension or bending. For both cases, two master nodes were used to control the node sets. There are two node sets called REPAIREDPANEL-1.LEFTEDGENODESET and REPAIREDPANEL- 1.RIGHT-EDGENODESET. The node sets represent the nodes on the edges left and right of the laminate, respectively.

In the case of tension, the boundary conditions are illustrated in Figure 60. On the left edge of the laminate, the master node RP_LEFT, also called RP-1, is completely constrained. The two displacement directions, which correspond to the displacement in the X direction and the Y direction, respectively, are U1 = 0 and U2 = 0. The rotational degree of freedom in the Z direction, which corresponds to UR3, is UR3 = 0; thus, the displacement/rotation in all 2D directions is constrained to 0. However, on the right edge, the master node RP_RIGHT, also called RP-2, is not completely constrained U1 $\neq$ 0 [m]. This non-zero displacement is the load that the laminate is subjected to. The other two degrees of freedom are fixed on the right edge, U2 = 0 and UR3 = 0.

In the case of bending, the boundary conditions are illustrated in Figure 61. The laminate constraints are the same as a simply supported beam. On the left edge, RP_LEFT, and the right edge, or RP_RIGHT, the two displacement degrees of freedom are U1 = 0 and U2 = 0. The rotational degrees of freedom in the Z direction on both nodes are UR3 $\neq$ 0. These non-zero rotations are the bending force that the laminate is subjected to. On RP_LEFT UR3 = $-\theta$ [rad], and on RP_RIGHT UR3 = θ. The angle θ is measured counter-clockwise.

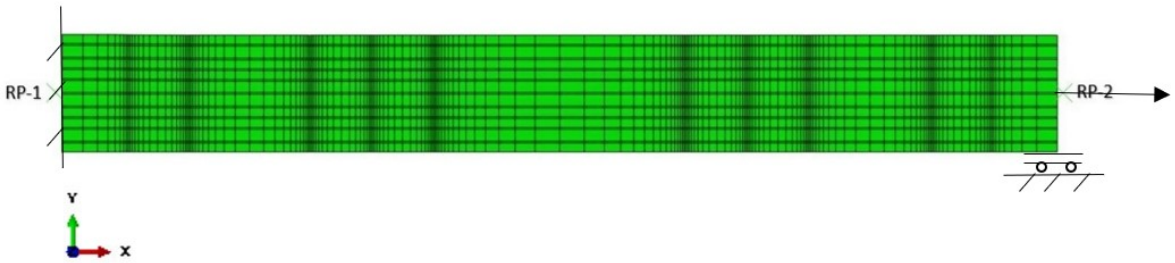


Figure 60: BC for tension

Figure 61: BC for bending

8 Stiffness functions

8.1 Equivalent stiffness function

In this function, the young's modulus of the laminate is calculated after the ABAQUS results have been read. To calculate the young's modulus the following equations have been used:

Variable	Value
$E_{laminate}$	Equivalent stiffness in (Gpa)
$RF1$	Reaction force on the right edge (N)
r	Length of the laminate (m)
$U1$	Displacement (mm)
t	Thickness of the laminate (m)

Table 4: Young's modulus variables

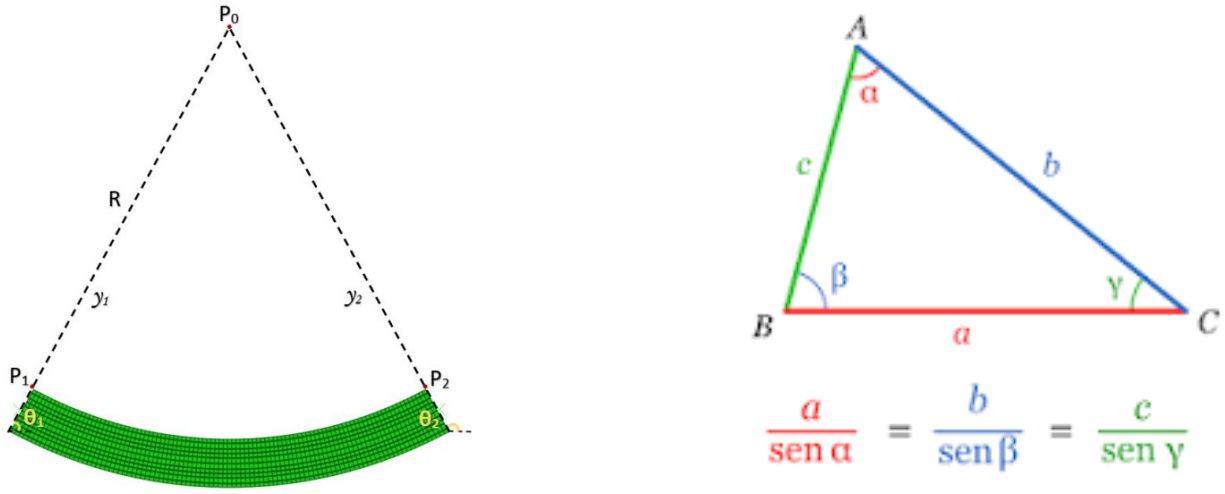
$$E_{laminate} = (F * L_0)/(\delta L * A) \tag{10}$$

$$E_{laminate} = (RF1 * r)/(U1 * t * 10^6) \tag{11}$$

Note: the area is the same number as the thickness of the laminate because the depth is 1m.

8.2 Bending stiffness function

In this function, the bending stiffness of the laminate is calculated after the ABAQUS results are read. The distance R which is the inverse of the curvature K_{rot} is calculated using the sine theorem in this triangle Figure 62



(a) ABAQUS triangle

(b) Theory

Figure 62: Sine therorem

Variable	Value
r	Length of the laminate (m)
R	c side of the triangle in 62b(m)
θ_2	Angle on the right edge of the laminate (rad)
$D_{laminate}$	Bending stiffness (Nm^2)
$RM3$	Reaction moment on the right edge (Nm)
$Krot$	Curvature of the laminate (m^-1)

Table 5: Bending stiffness variables

$$R = r * sin(\theta_2)/sin(\pi - 2 * \theta_2) \tag{12}$$

$$K_{rot} = 1/R \tag{13}$$

$$D_{laminate} = (RM3)/(K_{rot}) \tag{14}$$

9 Sustainable Development Goals

This project is aligned with the 7^{th} objective "Ensure access to affordable, reliable, sustainable, and modern energy." and with the 13^{th} objective "Take urgent action to combat climate change and its impacts." The study is a small contribution to a more competitive wind energy industry, and as a result, more affordable and clean energy (7^{th} goal). By using wind energy instead of oil or gas, CO2 emissions are reduced (13^{th} goal).

(a) 7^{th} sustainable development goal. (b) 13^{th} sustainable development goal.

Figure 63: Sustainable development goals [18]

9.0.1 7^{th} Affordable and Clean Energy

Between 2000 and 2018, the number of people with electricity increased from 78 to 90 percent, and the numbers without electricity dipped to 789 million.[18]

The population is still growing and more intensely in the poorest countries in the world. These countries need cheap energy to develop and for their population. The production of electricity using coal and gas is a typical way of getting this energy. The coal and gas based energy production was cheaper than the renewable energies when installed so it was the best option for the developing countries. This fact has contributed to the increment of the pollution in the air and the climate change.

The investment in renewable energies in Africa has been lower than the rest of the world. Only 2% of the world renewable energy investment of last 2 decades has been done in Africa [2]. Only 3% of the jobs created by the industry have been in Africa [2]. In addition, in Sub-Saharan Africa, electrification rate was static at 46% in 2019 with 906 million people still lacking access to clean cooking fuels and technologies [2].

Investing in solar, wind and thermal power, improving energy productivity, and ensuring energy for all is vital if we are to achieve SDG 7 by 2030 [18]. Expanding infrastructure and upgrading technology to provide clean and more efficient energy in all countries will encourage growth and help the environment. [18]

The creation of this project may be of help of for these countries. The program developed increases the optimization of the repairs in the wind turbines. The program also decreases

the amount of money spent in this type of repair since the simulations take less human resources. This contributes to a more competitive wind energy industry and reduces their maintenance cost making it more affordable for developing countries.

9.0.2 13^{th} Climate Action

The greenhouse effect is a problem because it's changing the climate. The global temperature is rising and more climate related disasters are happening. Between 1998 and 2017 1.3 million people have been killed, and left 4.4 billion injured by this climate related disasters [18].

Greenhouse gases emissions are higher by more than 50% of what they were in the 1990 [18]. To remedy this, the goal is to invest US$100 billion annually. This money is needed in developing countries to both adapt to climate change and invest in low-carbon energies [18].

This project will help in the low carbon energies objective by improving the life cycle of the wind turbines. This life cycle will be higher because the repairs will be better, and as a result, the blades will last longer and the LCOE will be reduced. The implementation of wind energy in the developing countries will help to decrease the annual C02 emissions, which will help reducing the climate change.

Overall, the program created can be a useful tool to help society battle the climate change and make the electricity more accessible for poor countries which are developing.

10 GUI code

In this section, the GUI code is presented. Only part of the code is in the appendix due to the size of the code. The whole code is uploaded into github.com. To access the code click on figure Figure 64. If the figure does not work the link is the following: `https://github.com/JorgeOT/MPSS-code`

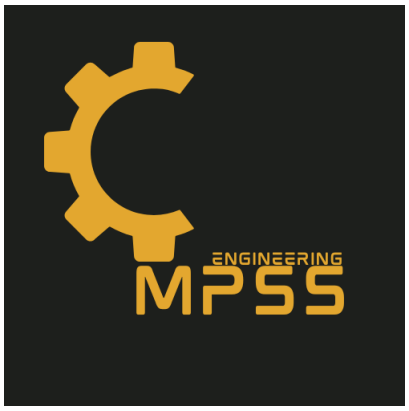


Figure 64: MPSS

11 Appendix D: Run and Read functions

11.1 Run function

In this section, the function that sends the input file to abaqus is presented.

```
1
2  function [RF1,RF2,RM3,U1] = Run_job_request_outputs_function(namefile,
       N_cores)
3  % close all
4  S = mfilename('fullpath');
5  f = filesep;
6  ind=strfind(S,f);
7  S1=S(1:ind(end)-1);
8  cd(S1)
9  % namefile = 'Case_A';
10 % N_cores = 8;
11 %above sets the path
12 namefileodb = '.odb';
13 delete(strcat(namefile,namefileodb));
14 namefilelck = '.lck';
15 delete(strcat(namefile,namefilelck));
16 namefileinp = '.inp';
17 pause(2) % can this pause stop the job from getting stuck?
18 stringsystem1 = strcat('abaqus job=',namefile);
19 stringsystem2 = strcat(' cpus=',num2str(cast(N_cores,"int8")));
20 stringsystem3 = strcat(stringsystem1,' inp=',namefile,namefileinp,
       stringsystem2,' interactive');
21 %prueba = stringsystem2
22 system(stringsystem3);
23
24
25
26 pause(2)
27 while exist(strcat(namefile,namefilelck),'file')==2
28     pause(0.1)
29 end
30
31 while exist(strcat(namefile,namefileodb),'file')==0
32     pause(0.1)
33 end
34
35 [RF1,RF2,RM3,U1]=Read_ODB_outputs_node_function(namefile);
36
```

```
end
}
```

11.2 Read function

In this section, the function that reads the output .odb file from ABAQUS is presented.

```
function [RF1,RF2,RM3,U1]=Read_ODB_outputs_node_function(namefile)
%clear all
%close all

debug = 0;
if (debug)
    debug_file = fopen('debug.dat', 'wt');
end
% namefile = CASE_A;
namefiledat = '.dat';
while exist(strcat(namefile,namefiledat),'file')==0
    pause(0.1)
end
file = strcat(namefile,namefiledat);
fidd = fopen(file);
i = 0;
pause(0.5)
numSteps = 0;
j=0;
while ( ~feof(fidd) )

    tline = fgetl(fidd);

    i = i+1;
    if (regexpi(tline, 'N O D E   O U T P U T')>0)  %For elements, replace '
       N O D E   O U T P U T'  by 'E L E M E N T   O U T P U T'
        numSteps = numSteps + 1;
        tline = fgetl(fidd);
        i = i+1;

        while(isempty(str2num(tline)))
            tline = fgetl(fidd);
            i = i+1;
        end
        while(~isempty(str2num(tline)))
            j=j+1;
            data_f = sscanf(tline, '%d %e %e %e %e', [1,5]);
```

```
37              if (debug)
38                  fprintf(debug_file, '%d\t%e\n', data_f(1), data_f(2), data_f
                        (3), data_f(4), data_f(5));
39              end
40              node_number=data_f(1);
41              tline = fgetl(fidd);
42              i = i+1;
43              RF1(j)=data_f(2);
44              RF2(j)=data_f(3);
45              RM3(j)=data_f(4);
46              U1(j)=data_f(5);
47
48          end
49          if (debug)
50              fprintf(debug_file, '\n');
51          end
52      end
53  end
54  if (debug)
55      fclose(debug_file);
56  end
57  fclose(fidd);
58  fclose all;
59  end
```

11.3 Resources

Two main tools are used to implement this project: MATLAB & ABAQUS. MATLAB is used to create the GUI and improve the MPSS algorithm. ABAQUS is used to simulate the repaired laminate. In MATLAB the GUI is designed in Appdesigner. The link between the MPSS and ABAQUS is done with a MATLAB function that sends and executes the .inp file in ABAQUS in parallel without having to open the program.

References

[1] "A robust and automated method for geometric modelling of thick laminates with multiple and asymmetric ply wrinkles". In: *Composites Part A: Applied Science and Manufacturing* vol. 141.106230 (2021), p. 106230. ISSN: 106230. DOI: https://backend.orbit.dtu.dk/ws/portalfiles/portal/269275892/1_s2.0_S0263822322001258_main.pdf.

[2] Irena International Renewable Energy Agency. *Renewable Energy Market Analysis: Africa and its Regions*. 2023. URL: https://www.irena.org/publications/2022/Jan/Renewable-Energy-Market-Analysis-Africa#:~:text=Africa20has%5C%20vast%5C%20resource%5C%20potential,and%5C%20other%5C%20low%5C%2Dcarbon%5C%20technologies..

[3] J. Beauson et al. "The complex end-of-life of wind turbine blades: A review of the European context". In: *Renewable and Sustainable Energy Reviews* 155 (2022), p. 111847. ISSN: 1364-0321. DOI: https://doi.org/10.1016/j.rser.2021.111847. URL: https://www.sciencedirect.com/science/article/pii/S136403212101114X.

[4] Mahdi Damghani Stephan Bolanos Amandeep Chahar Jason Matthews Gary A. Atkinson Adrian Murphy Timothy Edwards. *Design novel quality check and experimental test of an original variable length stepped scarf repair scheme*. 2022.

[5] Rufail L Laurin JJ Moupfouma F. "Composite aircraft lightning strike protection." In: *EUCAPAP* (2017), pp. 1–4.

[6] Michelle Froese. *What is lap shear strength in an epoxy?* 2020. URL: https://fastenerengineering.com/what-is-lap-shear-strength-in-an-epoxy/ (visited on 01/29/2023).

[7] GWEC. *GWEC | GLOBAL WIND REPORT 2022*. 2022. URL: https://gwec.net/wp-content/uploads/2022/04/Annual-Wind-Report-2022_screen_final_April.pdf (visited on 01/05/2023).

[8] Truong V-H Kwak B-S Roy R Kweon J-H. "Cohesive zone method for failure analysis of scarf patch-repaired composite laminates under bending load". In: *Compos Struct* (2019), p. 222.

[9] Yoo J-S Truong V-H Park M-Y Choi J-H Kweon J-H. *Parametric study on static and fatigue strength recovery of scarf-patch-repaired composite laminates.* 2016. URL: https://doi.org/10.1016/j.compstruct.2015.12.041. (visited on 11/29/2022).

[10] Forbes Julie Colemant. *Germany Reopens Coal Plants Because Of Reduced Russian Energy*. 2022. URL: https://www.forbes.com/sites/juliecoleman/2022/07/08/germany-reopens-coal-plants-because-of-reduced-russian-energy/ (visited on 01/05/2023).

[11] WEKA Industrie Medien. *Making Cement by Recycling Old Wind Turbine Blades*. 2022. URL: https://waste-management-world.com/artikel/making-cement-by-recycling-old-wind-turbine-blades/ (visited on 01/05/2023).

[12] Navid Mehreganian and Arash Soleiman Fallah. "10 - Blast loading effects on aircraft fuselage". In: *Multiphysics Simulations in Automotive and Aerospace Applications*. Ed. by Mojtaba Moatamedi, Thurai Rahulan, and Hassan Khawaja. Multiphysics: Ad-

vances and Applications. Academic Press, 2021, pp. 239–285. ISBN: 978-0-12-817899-7. DOI: https://doi.org/10.1016/B978-0-12-817899-7.00004-6. URL: https://www.sciencedirect.com/science/article/pii/B9780128178997000046.

[13] Aura Venessa De Guzman Paguagan. "Finite element modelling of repaired composite laminates". In: *DTU* (2022), pp. 1–207.

[14] University of South Australia Professor Peter Majewski Future Industries Institute. *How to manage future waste from wind turbine blades*. 2022. URL: https://www.energymagazine.com.au/how-to-manage-future-waste-from-wind-turbine-blades/ (visited on 11/29/2022).

[15] "Proof of a composite repair concept for aeronautical structures: a simplified method". In: *Mechanics Industry* Volume 20.8 (2020), p. 14. ISSN: 812. DOI: https://www.mechanics-industry.org/articles/meca/full_html/2019/08/mi190307/mi190307.html.

[16] Ole Sigmund and Kim Lau Nielsen. *Virtual Work Principle (Day 7)*. University Lecture. 2021.

[17] Michael Smith. *ABAQUS/Standard User's Manual, Version 6.9*. English. United States: Dassault Systèmes Simulia Corp, 2009.

[18] UN UN. *Sustainable development goals*. 2023. URL: https://www.undp.org/sustainable-development-goals?utm_source=EN&utm_medium=GSR&utm_content=US_UNDP_PaidSearch_Brand_English&utm_campaign=CENTRAL&c_src=CENTRAL&c_src2=GSR&gclid=CjwKCAjwq-WgBhBMEiwAzKSH6Ppjz4P1TOkTzDa9FEdVfnG_f56lLLkIT9axufZgq3hFiblDHKmmBwE.

[19] Sophie Vorrath. *Wind power and solar energy costs have increased, but much less than those of fossil fuels*. 2016. URL: https://www.evwind.es/2022/07/01/wind-power-and-solar-energy-costs-have-increased-but-much-less-than-those-of-fossil-fuels/86798 (visited on 11/29/2022).

[20] Liu B Xu F Feng W Yan R Xie W. *Experiment and design methods of composite scarf repair for primary-load bearing structures*. 2016. URL: https://doi.org/10.1016/j.compositesa.2016.05.011. (visited on 11/29/2022).

[21] Qing Han Bin Liu Wei Xie. "The tension failure mechanisms of composite stepped bonding repairs and joints for the aircraft primary load-bearing structures". In: *Journal of Adhesion Science and Technology* (2019), pp. 1–17.