

Danmarks  
Tekniske  
Universitet



---

# Multi-parameter step scarf repair graphical user interface documentation

---

**AUTHOR**

Jorge Ortega (s212758@dtu.dk)

February 10, 2023

# 1 GUI documentation

## 1.1 General Input

### 1.1.1 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.

### 1.1.2 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.

### 1.1.3 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 1.

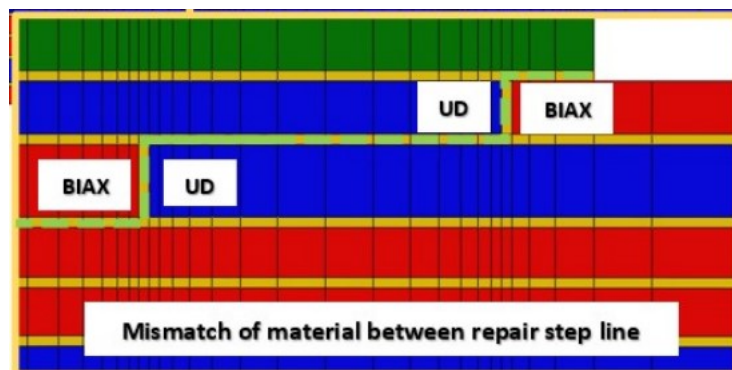


Figure 1: Mismatch representation [2]

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.[1]

### 1.1.4 Width defect

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

### 1.1.5 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.

### 1.1.6 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:

$$y = \frac{\sqrt{x_{curve}L}}{\sqrt{a_{curve}}}$$

Figure 2: Curvature formula

### 1.1.7 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.

### 1.1.8 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 pin-points. 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.

### 1.1.9 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.

### 1.1.10 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.

## 1.2 Geometry

### 1.2.1 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.

### 1.2.2 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.

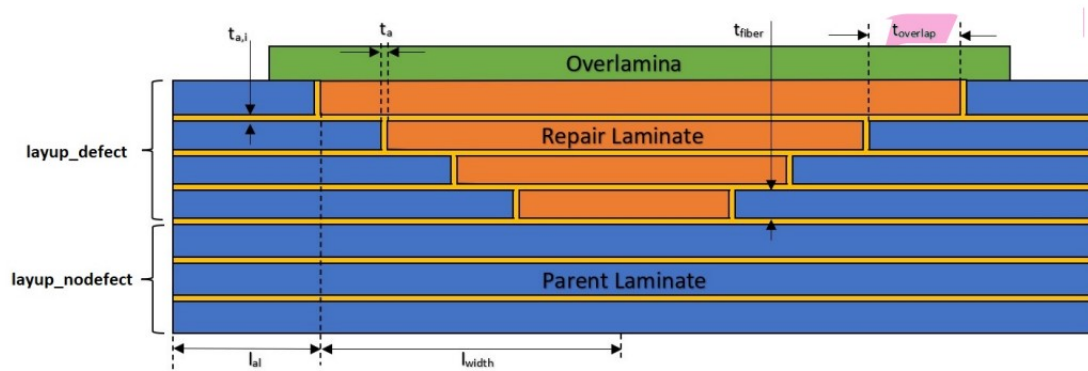


Figure 3: Overlapping layer length [2]

### 1.2.3 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 ( $t_{Hres}$ ). The thickness of the UD ( $t_{ud}$ ) and BIAx ( $t_{biax}$ ) 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 ( $t_{Vres}$ ). The thickness of the overlamina should be 'ud' or 'biax'. If the overlamina layer is not added, this value will not be used.

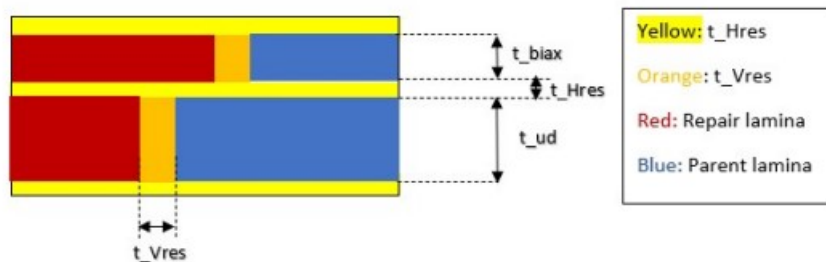


Figure 4: Different thicknesses [2]

### 1.2.4 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.

### 1.2.5 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:  $N^{\circ}zones = 3 * 2 + 1 = 7$  The general equation is the following:

$$N^{\circ}zones = N^{\circ}plies * 2 + 1 \quad (1)$$

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

### 1.2.6 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.

## 1.3 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.

## 1.4 Cohesive material properties, General Static Analysis and Step control

### 1.4.1 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.

### 1.4.2 Cohesive elements behavior

If cohesive element option is chosen, the damage evolution criterion that will be used is Benzeggagh-kane. The user inputs  $\eta$ . The following figure explains the behavior. Figure 5:

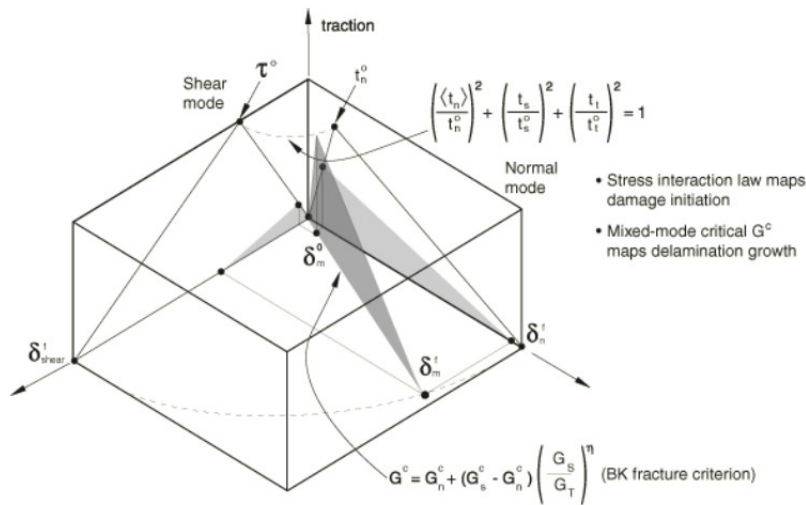


Figure 5: Mix mode response in cohesive elements [3]

### 1.4.3 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 2 reaches 1 the damage initiation limit is reached and the damage starts.

$$\left\{ \frac{\langle t_n \rangle}{t_n^o} \right\}^2 + \left\{ \frac{t_s}{t_s^o} \right\}^2 + \left\{ \frac{t_t}{t_t^o} \right\}^2 = 1 \quad (2)$$

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 3:

$$\max \left\{ \frac{\langle t_n \rangle}{t_n^o}, \frac{t_s}{t_s^o}, \frac{t_t}{t_t^o} \right\} = 1 \quad (3)$$

#### 1.4.4 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 6. 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.

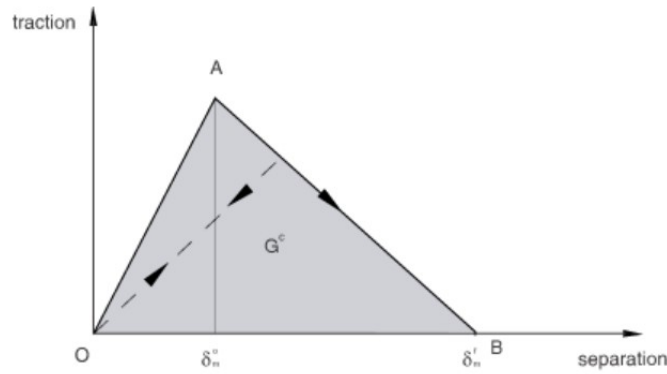


Figure 6: Fracture energy [3]

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 ([3]):

$$\begin{aligned}
 t_n &= \begin{cases} (1-D)\bar{t}_n, & \bar{t}_n \geq 0 \\ \bar{t}_n, & \text{otherwise (no damage to compressive stiffness)} \end{cases} \\
 t_s &= (1-D)\bar{t}_s \\
 t_t &= (1-D)\bar{t}_t
 \end{aligned} \tag{4}$$

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 [3].

#### 1.4.5 Linear elastic traction-separation behavior

In this box, the user can introduce the values for the elastic behavior of the resin element.[3]. 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.



$$\begin{Bmatrix} t_n \\ t_s \\ t_t \end{Bmatrix} = \begin{bmatrix} E_{nn} & & \\ & E_{ss} & \\ & & E_{tt} \end{bmatrix} \begin{Bmatrix} \varepsilon_n \\ \varepsilon_s \\ \varepsilon_t \end{Bmatrix} \quad (5)$$

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  $\varepsilon_n$ ,  $\varepsilon_s$ , and  $\varepsilon_t$  represent the corresponding nominal strains.

#### 1.4.6 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

#### 1.4.7 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. [3]

## 1.5 Run/Output

### 1.5.1 MENU 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 7

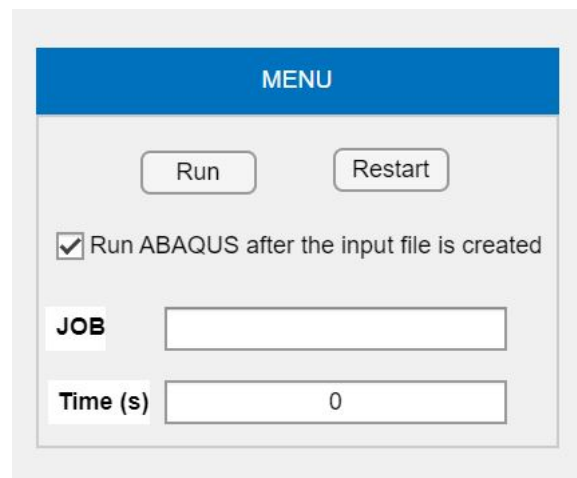
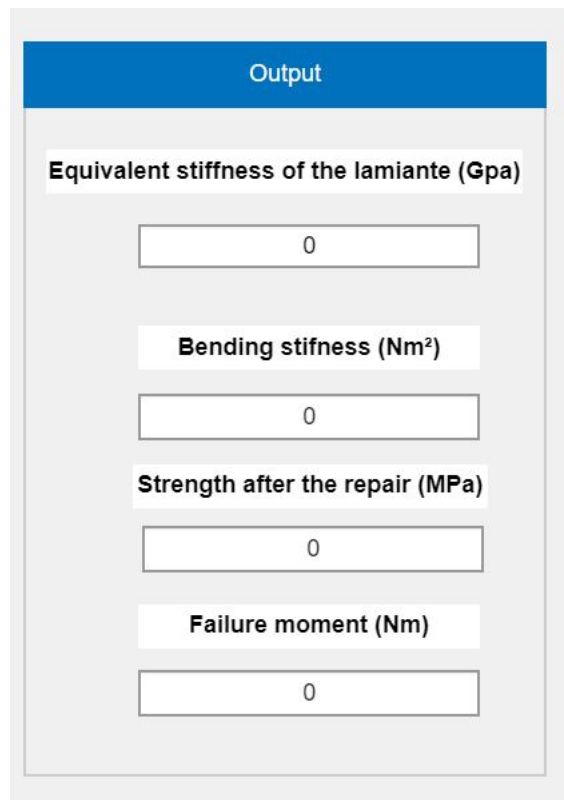


Figure 7: GUI MENU



The screenshot shows a software interface with a blue header bar labeled "Output". Below the header, there are five input fields, each preceded by a label in a light gray box. The labels and their corresponding values are:

- Equivalent stiffness of the lamiate (Gpa)**: 0
- Bending stiffness (Nm<sup>2</sup>)**: 0
- Strength after the repair (MPa)**: 0
- Failure moment (Nm)**: 0

Figure 8: 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 discretizing density is smaller than the sum of element number vector.

### 1.5.2 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 9) until the code has finished running. If the user wants to see the Load vs. Displacement curve, the engineer should select the 8<sup>th</sup> option in the Choose plot drop-down. The other plots are the steps the MPSS takes to create the repaired laminate.

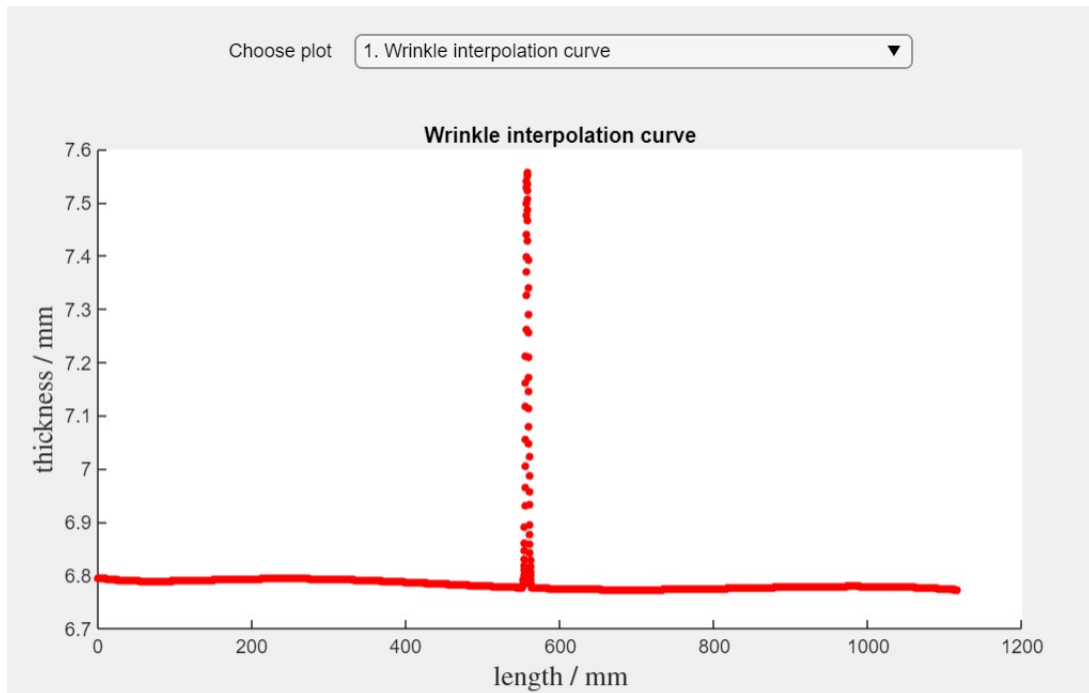


Figure 9: MPSS Plot

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.

## 1.6 Output: Stress and Strain

In this section Figure 15, 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.

## 1.7 GUI interface

## 1.8 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 22

## 1.9 GUI sections

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

The screenshot displays the 'General Input' tab of a MATLAB Appdesigner GUI. The interface is organized into several sections with blue headers:

- Input File name:** Includes a 'File name' field with 'CASE1.xlsx' and a 'Sheet' field with '1'.
- Output File name:** Includes a 'File name' field with 'CASE\_A'.
- Loading:** Features a 'Loading' dropdown set to '0: Tension', a 'Value' field, 'U1 Displacement in X (mm)' set to '10', and 'UR3 rotational displacement [rad]' set to '0.1745'.
- Ply by ply match:** Includes a 'Ply match' dropdown set to '1: ply by ply mis...'.
- Width of defect:** Includes a 'Width of the defect (mm)' field set to '1'.
- Overlamina Addition on top of repair:** Includes 'Overlamina' and 'Overlapping of the overlamina' dropdowns, both set to 'No'.
- Gaussian parameters guess:** Includes 'amplitude' (8), 'sigma' (8), and 'centre' (250) fields.
- Curve panel:** Includes a 'Curve' dropdown set to 'No' and a 'Parabola curve' field set to '0'.
- Resin Element:** Includes a 'Type of element' dropdown set to '1: Cohesive elements'.
- Number of CPU cores used for the simulation:** Includes a 'CPU cores' field set to '16'.

Figure 10: General input

In window Figure 10, the user inputs the general data and specifications of the desired laminate.

General Input	Geometry	Material properties	Cohesive material properties & General Static Analysis & Step control	Run/Output	Output: Stress & Strain	Guide 1	Guide 2
<div style="text-align: center; background-color: #0070C0; color: white; padding: 5px;">Layout Selection Process</div> <div style="display: flex; justify-content: space-between; align-items: flex-start; padding: 10px;"> <div> Flag: <input type="text" value="0: Composite with defect and layup contains two different lamin..."/>   Parent material: <input type="text" value="0: with the same material in repair and parent"/> </div> <div> Layup with defect e.g. ud,bi,ax,ud  <input type="text" value="bi,ax,bi,ax,ud,ud"/> </div> <div> From top to bottom  Layup without defect e.g. ud,bi,ax,ud  <input type="text" value="ud,ud,bi,ax,bi,ax"/> </div> </div>							
<div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">Overlapping layer length</div> <div style="padding: 10px;"> BIAx overlap (mm) <input type="text" value="48"/>  UD overlap (mm) <input type="text" value="48"/>  Overlamina overlap (mm) <input type="text" value="48"/> </div>				<div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">Geometry of 2D model</div> <div style="padding: 10px;"> Width from middle of defect (mm) <input type="text" value="250"/>  Additional allowance at the edges (mm) <input type="text" value="50"/> </div>			
<div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">Define material thickness</div> <div style="padding: 10px;"> Thickness of resin interface in between layers <input type="text" value="0.08"/>  Thickness of UD fibre lamina <input type="text" value="0.8"/>  Thickness of BIAx fibre lamina <input type="text" value="0.7999"/>  Thickness of resin in between repair and parent laminate <input type="text" value="0.08"/>  Thickness of the overlamina (bi,ax or ud) <input type="text" value="bi,ax"/> </div>				<div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">Discretising density</div> <div style="padding: 10px;"> Discretising density <input type="text" value="4000"/> </div>			
				<div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">Element number</div> <div style="padding: 10px;"> <p style="font-size: small;">It represents the grid size of each area in which the repair is divided into. e.g. Layup = ud,bi,ax,ud,bi,ax,ud,bi,ax. Width vector = 80,45,65,45,45,45,80</p> Width vector <input type="text" value="150,150,150,150,150,150,150,150"/> </div>			

Figure 11: Geometry

In window Figure 11, the user inputs the geometry of the desired laminate.

General Input	Geometry	Material properties	Cohesive material properties & General Static Analysis & Step control	Run/Output	Output: Stress & Strain	Guide 1	Guide 2																																																																								
<div style="display: flex; justify-content: space-between;"> <div style="width: 24%;"> <div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">UD material properties</div> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr><td>E1 (Pa)</td><td><input type="text" value="4.163e+10"/></td></tr> <tr><td>E2 (Pa)</td><td><input type="text" value="1.493e+10"/></td></tr> <tr><td>E3 (Pa)</td><td><input type="text" value="1.343e+10"/></td></tr> <tr><td>V12</td><td><input type="text" value="0.241"/></td></tr> <tr><td>V13</td><td><input type="text" value="0.2675"/></td></tr> <tr><td>V23</td><td><input type="text" value="0.3301"/></td></tr> <tr><td>G12(Pa)</td><td><input type="text" value="5.047e+09"/></td></tr> <tr><td>G13(Pa)</td><td><input type="text" value="5.047e+09"/></td></tr> <tr><td>G23(Pa)</td><td><input type="text" value="5.047e+09"/></td></tr> </table> </div> <div style="width: 24%;"> <div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">BIAx material properties</div> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr><td>E1 (Pa)</td><td><input type="text" value="1.392e+10"/></td></tr> <tr><td>E2 (Pa)</td><td><input type="text" value="1.392e+10"/></td></tr> <tr><td>E3 (Pa)</td><td><input type="text" value="1.392e+10"/></td></tr> <tr><td>V12</td><td><input type="text" value="0.53"/></td></tr> <tr><td>V13</td><td><input type="text" value="0.19"/></td></tr> <tr><td>V23</td><td><input type="text" value="0.02"/></td></tr> <tr><td>G12(Pa)</td><td><input type="text" value="1.15e+10"/></td></tr> <tr><td>G13(Pa)</td><td><input type="text" value="1.15e+10"/></td></tr> <tr><td>G23(Pa)</td><td><input type="text" value="4.539e+09"/></td></tr> </table> </div> <div style="width: 24%;"> <div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">UD material properties (different material)</div> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr><td>E1 (Pa)</td><td><input type="text" value="3.9e+10"/></td></tr> <tr><td>E2 (Pa)</td><td><input type="text" value="1.3e+10"/></td></tr> <tr><td>E3 (Pa)</td><td><input type="text" value="1.3e+10"/></td></tr> <tr><td>V12</td><td><input type="text" value="0.205"/></td></tr> <tr><td>V13</td><td><input type="text" value="0.205"/></td></tr> <tr><td>V23</td><td><input type="text" value="0.349"/></td></tr> <tr><td>G12(Pa)</td><td><input type="text" value="5.4e+09"/></td></tr> <tr><td>G13(Pa)</td><td><input type="text" value="3.4e+09"/></td></tr> <tr><td>G23(Pa)</td><td><input type="text" value="3.4e+09"/></td></tr> </table> </div> <div style="width: 24%;"> <div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">BIAx material properties (different material)</div> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr><td>E1 (Pa)</td><td><input type="text" value="1.25e+10"/></td></tr> <tr><td>E2 (Pa)</td><td><input type="text" value="1.25e+10"/></td></tr> <tr><td>E3 (Pa)</td><td><input type="text" value="1.25e+10"/></td></tr> <tr><td>V12</td><td><input type="text" value="0.498"/></td></tr> <tr><td>V13</td><td><input type="text" value="0.498"/></td></tr> <tr><td>V23</td><td><input type="text" value="0.498"/></td></tr> <tr><td>G12(Pa)</td><td><input type="text" value="1.03e+10"/></td></tr> <tr><td>G13(Pa)</td><td><input type="text" value="1.03e+10"/></td></tr> <tr><td>G23(Pa)</td><td><input type="text" value="1.03e+10"/></td></tr> </table> </div> </div>								E1 (Pa)	<input type="text" value="4.163e+10"/>	E2 (Pa)	<input type="text" value="1.493e+10"/>	E3 (Pa)	<input type="text" value="1.343e+10"/>	V12	<input type="text" value="0.241"/>	V13	<input type="text" value="0.2675"/>	V23	<input type="text" value="0.3301"/>	G12(Pa)	<input type="text" value="5.047e+09"/>	G13(Pa)	<input type="text" value="5.047e+09"/>	G23(Pa)	<input type="text" value="5.047e+09"/>	E1 (Pa)	<input type="text" value="1.392e+10"/>	E2 (Pa)	<input type="text" value="1.392e+10"/>	E3 (Pa)	<input type="text" value="1.392e+10"/>	V12	<input type="text" value="0.53"/>	V13	<input type="text" value="0.19"/>	V23	<input type="text" value="0.02"/>	G12(Pa)	<input type="text" value="1.15e+10"/>	G13(Pa)	<input type="text" value="1.15e+10"/>	G23(Pa)	<input type="text" value="4.539e+09"/>	E1 (Pa)	<input type="text" value="3.9e+10"/>	E2 (Pa)	<input type="text" value="1.3e+10"/>	E3 (Pa)	<input type="text" value="1.3e+10"/>	V12	<input type="text" value="0.205"/>	V13	<input type="text" value="0.205"/>	V23	<input type="text" value="0.349"/>	G12(Pa)	<input type="text" value="5.4e+09"/>	G13(Pa)	<input type="text" value="3.4e+09"/>	G23(Pa)	<input type="text" value="3.4e+09"/>	E1 (Pa)	<input type="text" value="1.25e+10"/>	E2 (Pa)	<input type="text" value="1.25e+10"/>	E3 (Pa)	<input type="text" value="1.25e+10"/>	V12	<input type="text" value="0.498"/>	V13	<input type="text" value="0.498"/>	V23	<input type="text" value="0.498"/>	G12(Pa)	<input type="text" value="1.03e+10"/>	G13(Pa)	<input type="text" value="1.03e+10"/>	G23(Pa)	<input type="text" value="1.03e+10"/>
E1 (Pa)	<input type="text" value="4.163e+10"/>																																																																														
E2 (Pa)	<input type="text" value="1.493e+10"/>																																																																														
E3 (Pa)	<input type="text" value="1.343e+10"/>																																																																														
V12	<input type="text" value="0.241"/>																																																																														
V13	<input type="text" value="0.2675"/>																																																																														
V23	<input type="text" value="0.3301"/>																																																																														
G12(Pa)	<input type="text" value="5.047e+09"/>																																																																														
G13(Pa)	<input type="text" value="5.047e+09"/>																																																																														
G23(Pa)	<input type="text" value="5.047e+09"/>																																																																														
E1 (Pa)	<input type="text" value="1.392e+10"/>																																																																														
E2 (Pa)	<input type="text" value="1.392e+10"/>																																																																														
E3 (Pa)	<input type="text" value="1.392e+10"/>																																																																														
V12	<input type="text" value="0.53"/>																																																																														
V13	<input type="text" value="0.19"/>																																																																														
V23	<input type="text" value="0.02"/>																																																																														
G12(Pa)	<input type="text" value="1.15e+10"/>																																																																														
G13(Pa)	<input type="text" value="1.15e+10"/>																																																																														
G23(Pa)	<input type="text" value="4.539e+09"/>																																																																														
E1 (Pa)	<input type="text" value="3.9e+10"/>																																																																														
E2 (Pa)	<input type="text" value="1.3e+10"/>																																																																														
E3 (Pa)	<input type="text" value="1.3e+10"/>																																																																														
V12	<input type="text" value="0.205"/>																																																																														
V13	<input type="text" value="0.205"/>																																																																														
V23	<input type="text" value="0.349"/>																																																																														
G12(Pa)	<input type="text" value="5.4e+09"/>																																																																														
G13(Pa)	<input type="text" value="3.4e+09"/>																																																																														
G23(Pa)	<input type="text" value="3.4e+09"/>																																																																														
E1 (Pa)	<input type="text" value="1.25e+10"/>																																																																														
E2 (Pa)	<input type="text" value="1.25e+10"/>																																																																														
E3 (Pa)	<input type="text" value="1.25e+10"/>																																																																														
V12	<input type="text" value="0.498"/>																																																																														
V13	<input type="text" value="0.498"/>																																																																														
V23	<input type="text" value="0.498"/>																																																																														
G12(Pa)	<input type="text" value="1.03e+10"/>																																																																														
G13(Pa)	<input type="text" value="1.03e+10"/>																																																																														
G23(Pa)	<input type="text" value="1.03e+10"/>																																																																														
<div style="background-color: #0070C0; color: white; padding: 5px; text-align: center;">Resin material properties</div> <div style="padding: 10px;"> E1 (Pa) <input type="text" value="3.241e+09"/>  V12 <input type="text" value="0.38"/>  Viscosity (Pa*s) <input type="text" value="1e-05"/> </div>																																																																															

Figure 12: Material properties

In window Figure 12, the user inputs the material properties of the desired laminate for UD, BIAx, and resin materials.

**General Static Analysis**

1st step (s): 0.1 Minimum time increment (s): 1e-25  
 Time period (s): 1 Maximum time increment (s): 0.1

**Damage initiation elastic limit (Pa). Criterion: Max stress & QUADS**

UD: 1e+09 BIAx: 1e+09  
 Resin 1: 7.5e+06 Resin 2: 1.5e+07 Resin 3: 1.5e+07  
 UD\_DIFF: 1e+09 BIAx\_DIFF: 1e+09

**Linear elastic traction-separation behavior**

Enn (Pa/m): 3e+14  
 Ess (Pa/m): 1.15e+14  
 Ett (Pa/m): 1.15e+14

**Damage evolution: Benzeggagh-Kenane**

UD: 2 BIAx: 2  
 Resin: 3.8  
 UD\_DIFF: 2 BIAx\_DIFF: 2

**Damage**

Damage stabilization: 0.01  
 Maximum SDEG: 0.95

**Damage evolution: Fracture energy (J/m<sup>2</sup>)**

UD 1: 100 UD 2: 100 UD 3: 100  
 BIAx 1: 100 BIAx 2: 100 BIAx 3: 100  
 Resin 1: 200 Resin 2: 1000 Resin 3: 1000  
 UD 1 DIFF: 100 UD 2 DIFF: 100 UD 3 DIFF: 100  
 BIAx 1 DIFF: 100 BIAx 2 DIFF: 100 BIAx 3 DIFF: 100

**Step control equilibrium: Time incrementation**

Io: 16 IR: 32

Figure 13: Cohesive material properties

In window Figure 13, the user input the cohesive material properties of the desired laminate for UD, BIAx, and resin materials.

Choose plot: 1. Wrinkle Interpolation curve

**MENU**

Run Restart  
☒ Run ABAQUS after the input file is created  
 JOB:   
 Time (s):

**Plot**

Y: 0 to 1  
 X: 0 to 1

**Output**

Equivalent stiffness of the laminate (Gpa):   
 Bending stiffness (Nm<sup>2</sup>):   
 Strength after the repair (MPa):   
 Failure moment (Nm):

Total thickness (mm):   
 Total length of the composite (mm):   
 Total length of the repair (mm):

Figure 14: Run and Output

In this window Figure 14, 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 15: Output: Stress and Strain

In this window Figure 15 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 16, Figure 17 and Figure 18 are a guide for the user to understand the variables correctly and to learn how to use the app.

General Input	Geometry	Material properties	Cohesive material properties	Run/Output	Output: Stress & Strain	Guide 1	Guide 2	+
<b>Input File name</b> In this box, the input of the wrinkle file name is needed. The input also needs to know the sheet of that excel file. See example: CASE 1.xlsx	<b>Define material thickness</b> Each of the elements in this box represents the thickness of the different layers. In the box 'thickness of the overlamina', the input should be: ud or biax.	<b>Output filename</b> In this box, the output file name is given. This will be the name of the .inp file that ABAQUS uses as input and the name of all the files that ABAQUS will give as output (.odb, .dat, .mdl, .msg, .prt, .res, .sim, .sta and .stt)	<b>Layup Selection Process</b> The layup defect ply order is written from top to bottom for both with and without defect. In this box, the type of process that the MPSS	<b>Overlapping layer length</b> It's the extra length of the new plys that overlap with the old layers below the repaired layer. It's the part where the newly repaired plys contact the old plys. See image:	<b>Loading</b> In this box, the type of loading (Tension or Bending) and their values should be given. The displacement X is along the UD fiber			

Figure 16: Guide 1

General Input	Geometry	Material properties	Cohesive material properties	Run/Output	Output: Stress & Strain	Guide 1	Guide 2	+
<b>Discretizing density</b> It represents the number of elements into which the part will be divided. It depends on the element size. This number should be bigger than the sum of the width vector elements to work: eg. 1000	<b>Resin Element</b> In this box, the user can choose between having a solid homogeneous or cohesive element for the resin layers.	<b>Ply by ply match</b> In this box, the user can choose between having a mismatch between the newly repaired plys and the old plys. The mismatch is done horizontally so BIAx doesn't touch BIAx and UD doesn't touch UD.	<b>Geometry of 2D model</b> In this box, the user can select the dimension of the 2D sample. Width from middle of defect [mm]. The additional allowance at the edges	<b>Element number</b> It represents the discretization distribution of the part. It represents the coarse to fine distribution. It should be higher where there is a change between the old ply and	<b>Gaussian parameters guess</b> They are used for the interpolation curve needed to make the wrinkle representation. It is important that the centre is around the centre of the	<b>Width of defect</b> It represents the width of the defect in the X direction.	<b>Curve panel</b> Allows the part to be curved. The parameter makes the parabola wider (lower a_curve) or narrower (higher a_curve) using the inverse parabolic function: $y = \frac{\sqrt{x_{curve}L}}{\sqrt{a_{curve}}}$ e.g. 0.5	
<b>Overlaminata Addition on top of repair</b> In this box, the user can choose to have or not an overlaminata on top of the repair and if that overlaminata has overlapping								

Figure 17: Guide 2

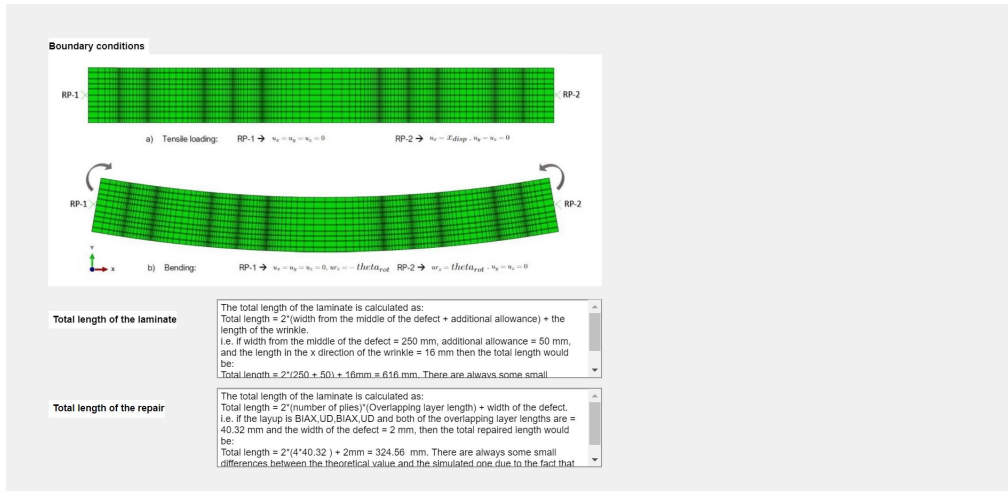


Figure 18: Guide 3

## 1.10 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 19. 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 20. 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  $UR3 = \theta$ . The angle  $\theta$  is measured counter-clockwise.

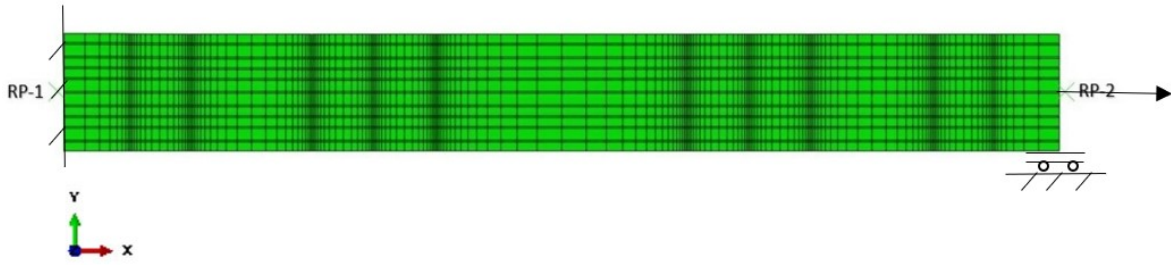


Figure 19: BC for tension



Figure 20: BC for bending

## 2 Stiffness functions

### 2.1 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 1: Young's modulus variables

$$E_{laminate} = (F * L_0) / (\delta L * A) \quad (6)$$

$$E_{laminate} = (RF1 * r) / (U1 * t * 10^6) \quad (7)$$

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

## 2.2 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 21

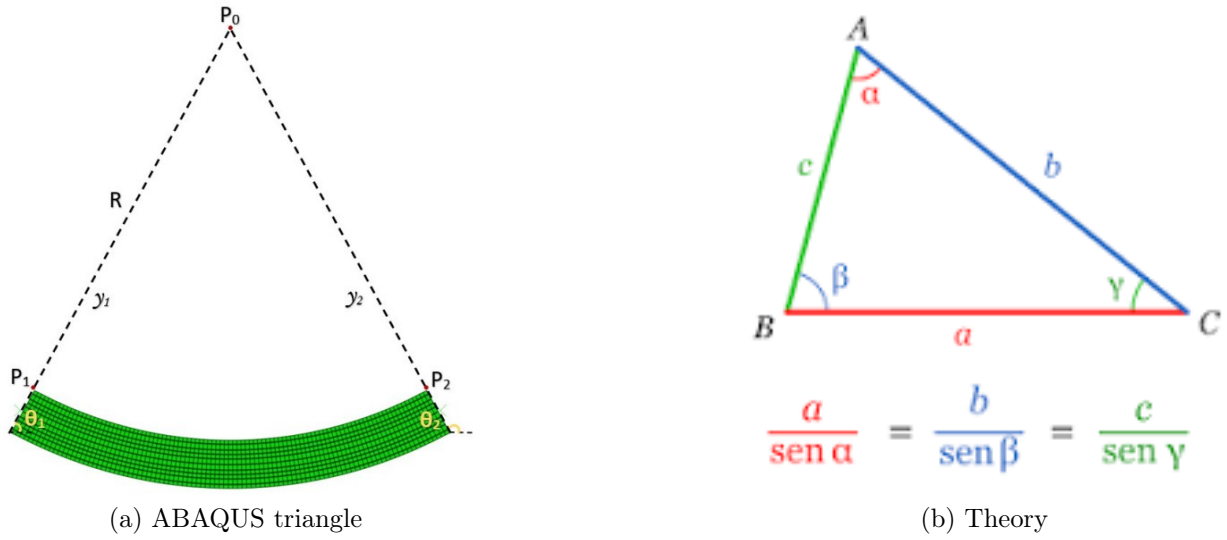


Figure 21: Sine theorem

Variable	Value
$r$	Length of the laminate ( $m$ )
$R$	c side of the triangle in 21b( $m$ )
$\theta_2$	Angle on the right edge of the laminate ( $rad$ )
$D_{laminat}$	Bending stiffness ( $Nm^2$ )
$RM3$	Reaction moment on the right edge (Nm)
$K_{rot}$	Curvature of the laminate ( $m^{-1}$ )

Table 2: Bending stiffness variables

$$R = r * \sin(\theta_2) / \sin(\pi - 2 * \theta_2) \quad (8)$$

$$K_{rot} = 1/R \quad (9)$$

$$D_{laminat} = (RM3)/(K_{rot}) \quad (10)$$

### 3 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 22. If the figure does not work the link is the following: <https://github.com/JorgeOT/MPSS-code>

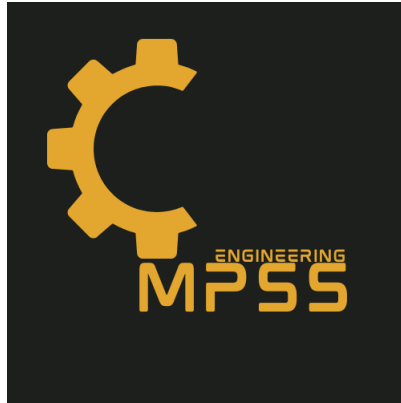


Figure 22: MPSS

### References

- [1] 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.
- [2] Aura Venessa De Guzman Paguagan. “Finite element modelling of repaired composite laminates”. In: *DTU* (2022), pp. 1–207.
- [3] Michael Smith. *ABAQUS/Standard User’s Manual, Version 6.9*. English. United States: Dassault Systèmes Simulia Corp, 2009.