Numerical Computation for Static Strength and Fatigue Life of Short Fiber Composites

Gabriel Mayr, and Dr. techn. Franz-Josef Falkner (supervisor)

Abstract—**The aim of this work is to establish an Ansys workflow for analyzing components composed of short fiber composites in terms of static strength and fatigue life. These materials exhibit anisotropic behavior, making simulations and calculations significantly more intricate compared to isotropic materials. The workflow commences with results from the injection molding simulation, which offers insights into fiber orientation. Fiber orientation plays a pivotal role in material stiffness and is defined through an orientation tensor for each discretized element. Ansys Material Designer is used to create a variable material comprising the matrix and fiber materials, dependent on the fiber orientation. Together, they define the orthotropic material properties of the composite. Subsequently, material properties are determined individually for each element in Ansys Mechanical, followed by a strength analysis. To assess static strength, a direction-dependent equivalent stress, accounting for anisotropy, is employed. Unlike the Von Mises stress, the Hill stress varies with direction and features distinct yield points for each orthotropic symmetry plane. Visualizing this Hill stress in Ansys Mechanical isn't feasible, so PyAnsys, a Python-Ansys interface for reading and transforming simulation data, is utilized. By utilizing the stress tensor in each node and the provided fiber orientation in each element, the Hill equivalent stress can be computed and visualized. To calculate fatigue life, the FEMFAT extension in Ansys is employed. Here, fatigue life is also influenced by fiber orientation. Depending on fiber orientation, a local Wöhler line is created at each node, and damage is calculated in conjunction with the applied stress. The node with the highest damage is deemed critical, impacting the component's fatigue life. This adaptable workflow can be applied to all components made of short fiber composites, offering deeper insights into the complexities of anisotropic materials during calculations.**

Index Terms—**Anisotropy, Fiber Orientation, Hill-Criterion, Fatigue**

Gabriel Mayr study at MCI, e-mail: g.mayr@mci4me.at.

I. Introduction

SHORT fiber composites are gaining increasing prominence across various industries. These materials are highly regarded for their combination of low density, robust strength properties, corrosion resistance, and streamlined production processes. Consequently, they are finding growing utility in diverse sectors. The customary production method for these composites involves injection molding, where short fibers are injected into the component alongside the matrix material. The precise positioning and injection angle have a significant impact on how these fibers are distributed within the component, a critical factor in determining material strength. Therefore, understanding the anticipated load on the component before the injection process is vital to achieving optimal strength. Nevertheless, short fiber composites pose a unique challenge due to their anisotropic behavior, stemming from the presence of these fibers. Unlike isotropic materials, which exhibit consistent material properties in all directions, short fiber composites display variations in material parameters along different axes. Consequently, applying traditional finite element analysis (FEA) to these materials becomes untenable. Hence, this work strives to develop an FEA-based approach for simulating anisotropic materials, considering the orientation of the fibers. The objective is to establish a workflow within Ansys that facilitates the computatuion and simulation of both static strength and component fatigue life for short fiber composite materials. In creating the workflow, emphasis is placed on ensuring efficiency and making it as straightforward as possible for users to apply.

Once the worfklow is set up, it is to be performed on the abutment, shown in Figure 1. The abutment is part of the steering lock installed in a STIHL lawn mower. This abutment is made of a short fiber composite material and is to be analyzed for static strength and fatigue life.

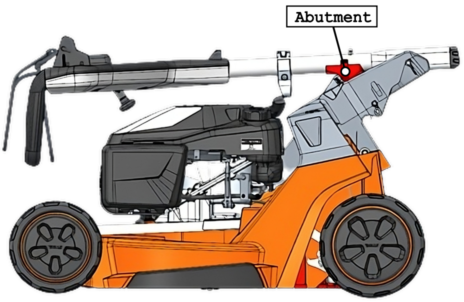

Fig. 1: Lawn mower with the abutment to be analyzed

II. Methods

A. Theory

To evaluate the strength of components, it is necessary to analyze the stresses to which they are subjected. These stresses are derived from the strains that appear in the material. The relationship between these two factors is illustrated by the stiffness tensor $[C]$, which is characterized by

$$[\sigma] = [C]\ [\epsilon] \tag{1}$$

and in known as the Hooke's law [1]. The stiffness tensor is a core factor in this work because it describes the material and its properties. For anisotropic materials, this tensor can be fully described with 21 independent variables. The more symmetries a material has, the fewer independent variables are needed to describe the stiffness tensor. For short fiber composites, where three symmetry planes are assumed, nine independent variable fully describe the tensor, and for isotropic materials only two. These independent variables are calculated with so-called engineering constants. These include Young's modulus, shear modulus and Poisson's ratio. Short fiber composites thus have different engineering constants in all three principal material directions and thus also different, direction-dependent material properties [2].

Short fiber composites do not have isotropic material behavior, since the fiber affects the stiffness more in the direction of the fiber than transverse to it. The matrix material has a lower stiffness than the fiber material. To calculate the stiffness of the composite, the material must be homogenized. The volume fraction ϕ of the fibers is an influential parameter. For unidirectional, continuous fibers, the stiffness tensor of the composite in the direction of the fibers is described with

$$\left[C^{UD}\right] = [C^m] + \phi\left([C^f] - [C^m]\right) \tag{2}$$

where the superscripts indicate the stiffness tensor of the fiber and the matrix, respectively. It is assumed that the strain of the matrix is equal to the strain of the fiber. Equation 2 can then be derived from Hooke's law. With a higher volume fraction of the fibers, the stiffness also increases. Equation 2 no longer applies to short fiber composites because the assumption of equal strains is no longer valid and there is no uniform parallel alignment of the fibers [3]. For short fiber composites, the orientation of the fibers is important for homogenization. In order to take the fiber orientation into account, it must also be mathematically describable. If a single fiber is considered, it can be described in a cartesian coordinate system with two angles (Figure 2).

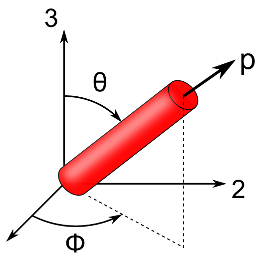

Fig. 2: Describing the orientation of a single fiber

The mathematical description of the vector p is equal to

$$p = \begin{Bmatrix} p1 \\ p2 \\ p3 \end{Bmatrix} = \begin{Bmatrix} \sin\theta\cos\phi \\ \sin\theta\sin\phi \\ \cos\theta \end{Bmatrix}. \qquad (3)$$

To better represent the orientation of the fiber, the dyadic product of the vector p is formed. As a result, a second order tensor is obtained and it is called orientation tensor A. To represent the orientation of several fibers, the average of the orientation tensor is calculated with

$$A = \frac{1}{N}\sum_{k=1}^{N}(p \otimes p)_k \qquad (4)$$

where N is the number of fibers and k the running variable. As an example, if two fibers are assumed to be parallel in the direction of the first coordinate axis, with angles ϕ equal to zero and θ equal to $\pi/2$, an orientation tensor of

$$A = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \qquad (5)$$

is obtained from equation (4). The orientation tensor has the properties that the eigenvectors are the material principal directions and the eigenvalues are the degree of alignment of the fibers in the respective principal direction. Additionally, the orientation tensor is symmetric and always has the trace equal to one [2]. The orientation tensor can also be determined with an orientation distribution function, but this will not be discussed in detail due to its scope. The averaged stiffness matrix for short fiber composites is then determined by multiplying the entries of the orientation tensor by the independent variables of the stiffness matrix of equation (2). The stiffness of the material is thus not only dependent on the volume fraction, but also on the orientation of the fibers [3].

B. *Material Properties in Ansys*

To be able to implement the orthotropic material properties in Ansys, the component systems Material Designer and Injection Molding Data are available in Ansys Workbench. The homogenization process is performed with the Material Designer. In this process, matrix and fiber material, as well as the volume fraction are determined and a variable material is created. Variable refers to the fact that the material is dependent on the orientation tensor. The orientation tensor for each individual element is the result of a previously performed injection molding simulation. With the help of the two component systems, the material properties are implemented in Ansys Mechanical. To show the implementation in this section, a simple reference example in the form of a cube with eight elements is used. In Figure 3, the element orientations are shown for each of the eight elements. These element orientations represent the principal material directions [4]. The orientation tensor file is used in this example from an Ansys Tutorial and is used for illustration purposes only.

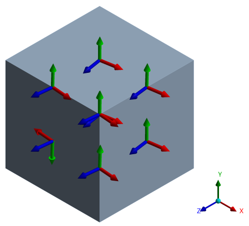

Fig. 3: Element orientations on the reference example

The material parameters in each element are determined. Figure 4 shows the Young's modules in the X-direction of the element coordinate systems [4].

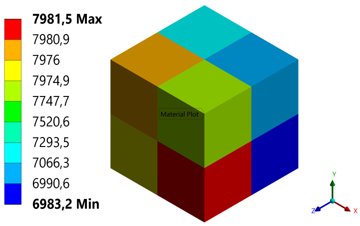

Fig. 4: Different Young's modules in each element

C. Static Strength Analysis

Stresses are evaluated to analyze the strength of components. What stress is used for the evaluation usually depends on the material. The most commonly used equivalent stress is the Von Mises equivalent stress. For the calculation of short fiber composites, however, the Von Mises stress is not suitable because it is direction-independent and does not take into account the fiber orientation. Therefore, the Hill equivalent stress is used for the calculation of short fiber composites. The Hill equivalent stress is direction-dependent and determines six different yield strengths for the material. There are three different yield strengths in the three principal directions and three for the different shear stresses. The ratios of the yield stress in each direction to the reference yield stress of the material are determined with coefficients. The coefficients are in turn dependent on the fiber orientation, the aspect ratio of the fiber and the volume fraction. Loads transverse to the fiber are thus weighted higher than loads parallel to the fiber direction. If the calculated total Hill equivalent stress $\hat{\sigma}$ is greater than the reference yield stress σ_y of the material, plastic deformation occurs [3]. The Hill criterion is thus calculated as

$$f(\sigma, \sigma_y) = \hat{\sigma} - \sigma_y = 0. \tag{6}$$

Since it is not possible to visualize the Hill equivalent stress in Ansys by default, a method to calculate it has to be found. For this purpose, PyAnsys, a newly implemented Python-Ansys interface, is used. With the help of PyAnsys simulation data can be read and transformed. This creates a variety of new possibilities for post-processing. The simulation data is read from the result file and thus receives all data that is also available in the Workbench. By reading the eigenvalues of the orientation tensor for each element, the Hill coefficients can be calculated for each individual element. With the element orientations output as Euler angles, the rotation matrix can be computed for each element, which accurately represents the material principal directions. By extracting the stress tensor in each node, together with calculated rotation matrix and the Hill coefficients, the Hill equivalent stress in each node can be analyzed. A big advantage of PyAnsys is that self generated results can be visualized and the calculated Hill equivalent stress can be displayed directly in Mechanical. Ansys functions, such as section planes, can thus be performed in the same way as for all implemented Ansys results. Figure 5 shows the Hill equivalent stress for the reference example, where a force of 1 kN is applied in the global X-direction.

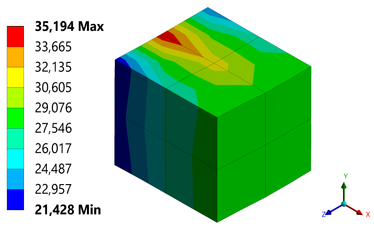

Fig. 5: Hill Stress calculated with PyAnsys

It is noticeable in Figure 5 that there is no homogeneous stress state in the cube due to a normal force in the global X-direction. This is because of the anisotropy, since the coupling in the stiffness matrix also results in shear stresses [1]. Due to the anisotropy, it is often possible that the maximum stresses occur inside the component. Due to the Mechanical function of the section planes, these interiors can also be evaluated on the self-generated result. The PyAnsys code that leads to the calculation and visualization of the Hill equivalent stress is applicable to any component made of short fiber composites. It is also possible to display the degree of utilization on the component. The utilization factor is calculated by relating the occurring Hill equivalent stresses to the reference yield point.

D. Fatigue Life Analysis

The second analysis included in the workflow is a fatigue life calculation. This calculation is performed in the Ansys extension FEMFAT. The import of the material properties is the same as for the static strength analysis, as the fatigue analysis is an extension of the static strength analysis. Before starting with the implementation, the calculation concept of FEMFAT is introduced. A Wöhler-line is used to evaluate the fatigue life. A schematic Wöhler-line with the three characteristics, durability strength σ_D, the number of cycles to durability N_D and slope k, is shown in Figure 6 [5].

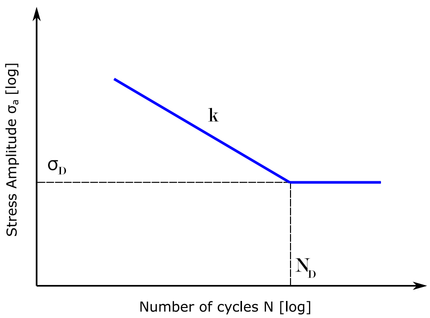

Fig. 6: Schematic Wöhler-line with its parameters

When a material is defined, a material Wöhler-line is created based on the mechanical properties. FEMFAT calculates with the influence parameter concept, which locally modifies the material Wöhler-line considering different influences. Thus, a separate local Wöhler-line is generated for each node of the finite element mesh. Influences such as surface roughness, component size or notch factor influence the three parameters durability strength, number of cycles to durability and slope. A damage accumulation according to Miner is then performed at each node and the damage at each node is calculated. The node with the maximum damage is considered critical [6]. By applying the critical section plane method, multiaxial and random loads can also be considered in FEMFAT. A damage analysis is performed locally in each section plane. The plane with maximum damage is considered critical for component failure and the corresponding damage value is assigned to the currently considered node as a result [6][7]. The presented calculation concept is extended for orthotropic materials, such as short fiber composites. Static material properties, such as Young's modulus, yield strength and tensile strength, along and transverse to the fiber are input data for the fatigue life calculation. To account for anisotropy, the existing section plane method is extended by using different Wöhler-lines in each plane depending on the position of the plane relative to the fiber orientation. Two interpolations are used to determine these Wöhler-lines. In the first interpolation, shown in Figure 7, the material parameters in the three principal anisotropy directions are required. The principal directions are taken together with the fiber fractions from the orientation tensor (eigenvectors and eigenvalues of the tensor). The local material parameters can be determined by linear interpolation from the two specimen tests. The symbol w stands for any material parameter, such as Young's modulus or yield strength. The symbols w_1, w_2 and w_3 are the interpolated and extrapolated parameters in the principal anisotropy directions, respectively [6].

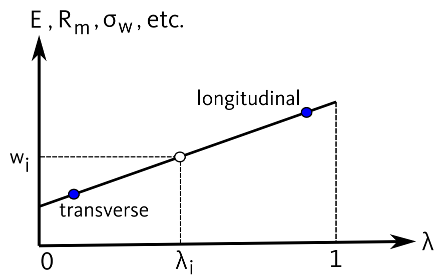

Fig. 7: First interpolation of material parameters

In the second interpolation, shown in Figure 8, the material parameters in the section plane, defined by the normal vector, are also calculated by interpolation. A sinusoidal variation of the material parameters is assumed if the section plane is rotated by 90 degrees from one principal anisotropy direction (e.g. e_1) to another (e.g. e_2) [6].

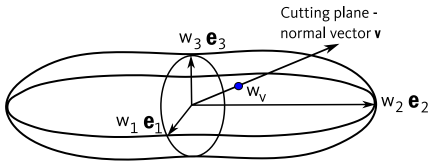

Fig. 8: Second interpolation of material parameters

To perform the calculation in FEMFAT inside Ansys, the first task to do is to create a channel. Stresses, together with a load-time history, form a channel [8]. The stresses are taken directly from the finite element calculation. The load time history indicates which load type with which mean stress and amplitude the stresses act on the component. In this case, only a load-time history is available from an Ansys Tutorial, which represents an alternating load with 20 load cycles. The mean stress is equal to zero and the amplitude factor is equal to ± 0.5. Due to the low load cycles and an amplitude of just ± 0.5, a minimal damage is to be expected. For the material definition, since version 4.6, FEMFAT offers a material generator for short fiber composites. With the material definition of matrix and fiber with the corresponding volume fraction, the material generator provides the static material parameters for the interpolation [9]. As mentioned, FEMFAT calculates with the influence parameter concept. The fiber orientation is also an influence parameter and must be transfered to FEMFAT in the form of an orientation tensor for each element. Ansys can map the fiber orientation tensor profile created in the injection molding simulation for each element to the mesh in Ansys Mechanical. However, FEMFAT does not have this feature and thus, a fiber orientation tensor file must be generated for the currently used mesh in Ansys Mechanical. This file has a separate orientation tensor for each element. Through the element orientations and the eigenvalues, the orientation tensor in each element can be calculated. By a script written in the course of this master thesis, a fiber orientation tensor file is created directly in Mechanical, which is adapted to the mesh in Mechanical. This created file has to be included in the calculation of FEMFAT in an XML format. Thus, all settings are made that are necessary to perform a fatigue life analysis. Figure 9 shows the damage on the reference example.

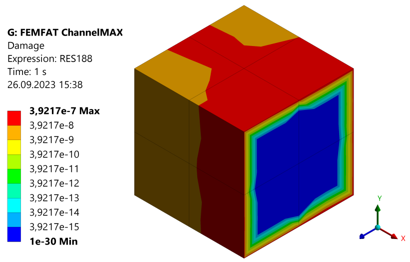

Fig. 9: Damage on the reference example

As expected, the damage in this example is minimal. This is due to the low voltages and low load cycles used for this example. Fatigue life calculations on the reference example, where in the first test, the fibers are aligned parallel to the loading direction and in the second test the fibers are aligned transversely to the loading direction, have shown that the damage in the second test is twelve times larger than in the first test. This demonstrates the huge influence of the fiber orientation.

III. Results

The presented workflow is designed to be applicable to any component made of short fiber composites. Therefore, the created workflow can also be performed on the abutment in the steering lock. Figure 10 shows the structure of the steering lock with four individual components. The limit stop has a fixed support and is therefore locked in all directions. A frictional contact prevails between the limit stop and the ratchet lever as well as between the handlebar and the abutment. In addition, the handlebar is screwed onto the abutment with a preload of 1 kN. The ratchet lever and the handlebar have a revolute joint in their holes. A moment of 79.25 Nm acts at the front end of the handlebar, which attempts to pull the handlebar out of the abutment. The pin of the ratchet lever hooks onto the abutment and should thus prevent upward movement. For the abutment, which is made of a short fiber composite, with the mentioned boundary conditions, a static strength analysis and a fatigue life analysis should be performed. Due to the boundary conditions, the maximum stress as well as the maximum damage to the abutment are to be expected in the lower area where the pin of the rachet lever engages.

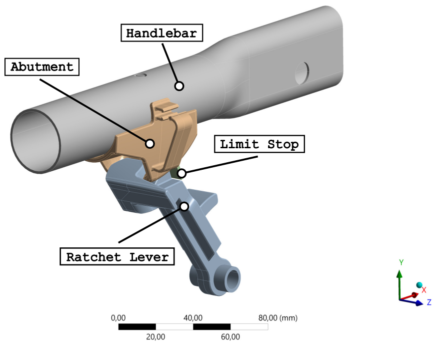

Fig. 10: Damage on the reference example

A. Results of the Static Analysis on the Abutment

With the fiber orientation tensor file resulting from the injection molding simulation and the material definition in Material Designer, the orthotropic material parameters for the abutment can be determined with Ansys. With the workflow created in this work is the implementation in Ansys efficiently possible. As a result, the utilization factor at the abutment is evaluated, based on the Hill equivalent stress (Figure 11). As expected, the maximum is at the bottom of the abutment, where the ratchet lever hooks into the abutment. With the load factor, the safety factor can be calculated under the given boundary conditions. It can be stated that the applied force acting on the abutment during transport of the lawn mower does not cause any plastic deformations in the static load case.

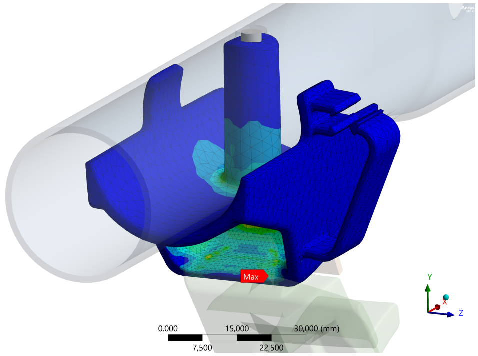

Fig. 11: Utilization rate on the abutment

B. Results of the Fatigue Analysis on the Abutment

The created workflow for calculating the fatigue life is also applied to the abutment. It is important to note that this fatigue analysis can only be performed with the load-time history of the Ansys Tutorial, since no real load history is available for the abutment. Due to the low number of oscillations and the low amplitude factor, the damage, shown in Figure 12, is minimal.

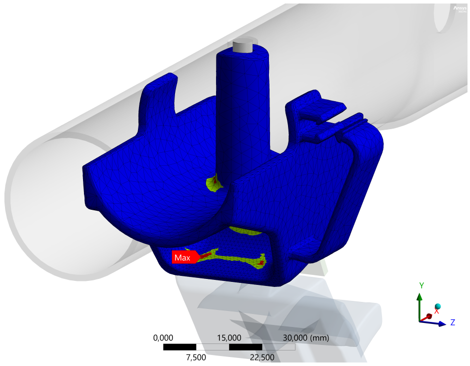

Fig. 12: Maximum damage on the abutment

The fatigue life calculation workflow works efficiently and can be applied to all components made of short fiber composites.

IV. Conclusion

As a result of the calculations in this master thesis, it can be stated that the set goals have been met and great expertise in the field of short fiber composites has been achieved. The anisotropic material behavior presents a great challenge, since many fundamental laws of engineering mechanics that apply to isotropic materials are no longer valid. In order to understand anisotropy, theoretical foundations around the stiffness matrix are inevitable. In the calculation of short fiber composites, the orientation tensor is the center point. The orientation tensor describes how the fibers are oriented in the component and thus determines the anisotropic material behavior and its material parameters. Ansys already offers many functions for the calculation of short fiber composites and enables simple implementation to determine the orthotropic material parameters. For the static strength analysis, PyAnsys opens up a variety of new options that are possible in an FE simulation. It was achieved that a direction-dependent equivalent stress can be calculated and visualized within Ansys. PyAnsys is a new feature in Ansys, which is developing dynamically. Also during the elaboration of this thesis new options have been introduced. It will be interesting to see what develops in this area in the next few years. In the case of the fatigue life analysis, the implementation of short fiber composites in Ansys is already well developed and documented. A helpful feature is the extension FEMFAT inside Ansys. Thus, it is not necessary to switch between two softwares for the calculation, the entire workflow can be carried out in Ansys. The mapping of the fiber orientation tensor file can also be carried out directly in Ansys with the PyAnsys script developed in this thesis. There also exist mapping softwares that can perform exact mapping of the orientation tensor to the current mesh, but with the code created in this work, it can be mapped and exported directly to Ansys with one simple click and the entire workflow can be implemented in Ansys. Overall, the fatigue life analysis is straightforward and quick to perform.

Acknowledgment

The authors would like to thank the company STIHL, in the person of Dr. Manfred Hofer, who made this Master's thesis possible. A constant exchange and queries were possible at all times. Furthermore, we would like to thank the staff of the structural dynamics laboratory at the MCI, with whom a professional exchange was always possible.

References

[1] H. Mang and G. Hofstetter, *Festigkeitslehre*, zweite, aktualisierte auflage ed., ser. Springer eBook Collection Computer Science and Engineering. Vienna and s.l.: Springer Vienna, 2004.

[2] C. L. Tucker, III, *Fundamentals of fiber orientation: Description, measurement and prediction*. Munich: Hanser Publishers, 2022.

[3] ANSYS Help, "Material Designer User's Guide: Release 2023." [Online]. Available: https://ansyshelp.ansys.com

[4] ——, "Short Fiber Composites Tutorials: Release 2023." [Online]. Available: https://ansyshelp.ansys.com

[5] Engineering Center Steyr GmbH & Co KG, "Theory Manual: FEMFAT." [Online]. Available: https://femfat.magna.com/download/documentation

[6] Hanser, "MP Materials Testing : Materialprüfung ; materials and components, technology and application," München.

[7] C. Gaier and H. Dannbauer, "Fatigue Analysis of Multiaxially Loaded Components with the FE-Postprocessor FEMFAT-Max," Austria.

[8] Magna International, "FEMFAT Version 2023 - ChannelMax TransMax." [Online]. Available: https://femfat.magna.com/download/documentation

[9] ——, "FEMFAT Documentation Release Notes Version 2023 ." [Online]. Available: https://femfat.magna.com/download/documentation

Gabriel Mayr is a Master student of Mechatronics and Smart Technologies at the MCI Innsbruck/Austria. In the context of his master's thesis, he worked on the simulation and calculation of short fiber composites.