

Consideration of Orientation Properties of Short Fiber Reinforced Polymers within Early Design Steps

Georg Gruber, Andreas Haimerl and Sandro Wartzack

Chair of Engineering Design KTMfk
University of Erlangen-Nuremberg FAU, Germany

Abstract

Within the modern automotive industry there is an increasing application of parts made of short fiber reinforced polymers (SFRP). The reasons are their beneficial mechanical properties and their series production capability. However, the prediction of their crash behavior by simulation is very complicated, since a precise simulation requires considering the fiber orientation distribution. That's why, in early design steps often only imprecise, isotropic simulation approaches are deployed in order to save calculation time and license costs for additional software tools.

The aim of the present paper is to introduce a simplified simulation approach allowing an anisotropic simulation taking into account the orientation data obtained by an injection molding simulation. To enable its application in early design steps only standard functions already implemented in LS-DYNA[®] are deployed. The complex material behavior of short fiber reinforced polymers is represented by overlapping two standard material models of LS-DYNA in one single shell definition. The input parameters of the resulting phenomenological material description are obtained by using optimization methods. The methodology being used to convert the orientation data in order to set up an executable input deck is supported by two self-developed software tools. The first software tool extracts the orientation angles from the process simulation by assigning fiber orientation tensors to corresponding shell elements of the mesh of the crash simulation. For each shell element the orientation data are averaged and projected on the shell. By doing so, the complex orientation state is reduced to just three values per shell element – one fiber orientation angle and two fiber orientation probability values. Based on these data, the second software tool creates the executable input deck. The legitimacy of the presented approach is proved by an experimental validation: SFRP-plates are analyzed within a drop weight test. Despite the mentioned simplification (reduction of the complexity of the orientation state) the numerical results show a strong correlation with the experimental data.

1. Introduction

Within the modern automotive industry there is an increasing application of parts made of short fiber reinforced polymers (SFRP). The reasons are their beneficial mechanical properties and their series production capability with help of the injection molding technique. However, the prediction of their crash behavior by simulation is very complicated due to their complex and manufacturing dependent anisotropic material behavior, which contradicts their application. To achieve accurate simulation results, it is mandatory to carry out an additional injection molding simulation and consider the derived fiber orientation distribution within the following structural simulation. In the literature this simulation method is referred to as *integrative simulation*.

Within early design steps when several different design proposals have to be evaluated, methods allowing a quick and efficient estimation of the mechanical properties are preferred by the product developer. Consequently in these phases the influence of reinforcement fibers is usually neglected and the Finite Element simulations are performed isotropically. By these isotropic simulations only results of minor accuracy can be obtained. Accordingly, the high degree of freedom of design available in early design steps cannot be exploited adequately. So within the first evaluation of different design proposals the optimum design cannot necessarily be discovered.

The aim of the present paper is to introduce a simulation method supporting the consideration of orientation properties of SFRPs determined within an injection molding simulation. The

simulation approach shall be easy to use enabling its application in early phases of the product development process.

The paper is structured as follows: At first the basic idea of integrative simulation approaches are described. Afterwards state of the art tools and approaches in the field of the integrative simulation will be discussed briefly. The paper's focus lies on the description of the approach of the Chair of Engineering Design (KTmfk), which is supported by two software tools. As validation a weight drop test will be performed by simulation and will be opposed corresponding experimental data. The paper is concluded with a short summary and discussion.

2. Fundamentals of the integrative simulation

To enable an accurate prediction of the mechanical behavior of SFRP-parts an integrative simulation has to be performed. Basically this type of simulation can be divided into two challenges. On the one hand, a numerical material description covering all relevant effects of SFRP-parts is required. Important effects to be mentioned in this context are non-linearity, plasticity, strain-rate dependency and anisotropic stiffness as well as strength behavior. On the other hand, the orientation state resulting from the manufacturing process has to be determined and the orientation data have to be prepared in order to be utilizable as input for the following anisotropic structural simulation.

The anisotropic material properties result from the fiber distribution in the final part, which in itself is a result of the flow process within the injection molding process. To put it simply, the fibers in the outer layers of a thin walled part are aligned along the fluid flow of the polymer. The fibers of the center layer of the part are aligned rather perpendicular to the flow direction (see fig. 1a). The fiber distribution across the whole part can be derived by injection molding simulation. The orientation condition can be described with a fiber orientation tensor at each node of the mesh of the process simulation. According to the definition of ADVANI/TUCKER [1] the fiber orientation can be described as a symmetric 3x3 matrix. By performing a principal axis transformation, the orientation distribution can be displayed as orientation ellipsoid, as displayed in fig. 1b. The eigenvectors e_i portray the principal direction of the fiber distribution (*orientation angle*), whereas the eigenvalues λ_i indicate the *orientation distribution probability* (ODP) of the corresponding principal axis. Consequently, the eigenvectors deliver the direction and the eigenvalues deliver the degree of the anisotropic material properties. The sum of all ODP values equals 1. Studies comparing the fiber orientation distribution derived by simulations with the orientation distribution obtained by image analysis of polished cross section show a strong correlation between simulation and measurement [2].

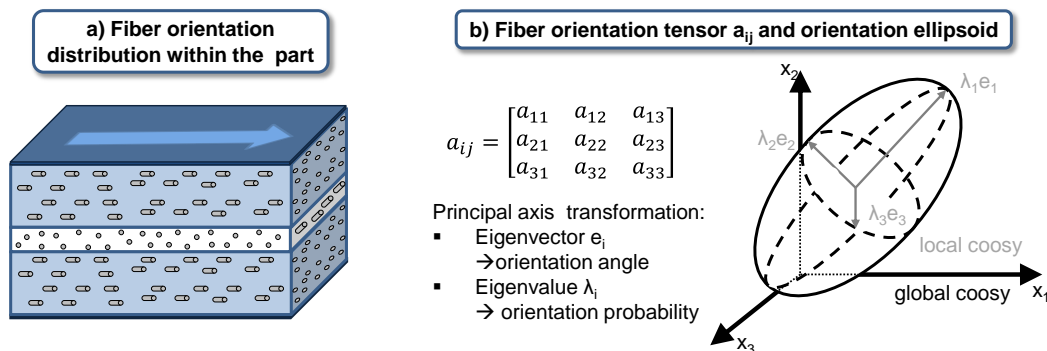


Figure 1: a) Fiber orientation distribution in a section of the thin walled part
b) Orientation tensor according to [1]

However, the orientation data of the injection molding simulation cannot be mapped directly on the mesh of a following structural simulation due to mesh inconsistency. Process simulation tools use tetrahedron meshes for the discretization, whereas within crash simulation of thin walled structures 2D quad shell elements are beneficial.

3. Existing tools and approaches of integrative simulation

The realization of an accurate simulation of SFRP-structures is an important task in the field of CAE for several years. There are already advanced commercial tools for the integrative simulation (e.g. [3], [4]), which have to be coupled with commercial FE-codes like LS-DYNA[®]. On the one hand, the integrative simulation applications realize the mapping of the orientation information. On the other hand, a material description allowing the consideration of all relevant effects of the material behavior is provided. The numerical input parameters for the corresponding material models are usually determined by reverse-engineering procedures of standard material tests. The tools mentioned in [3] and [4] enable the prediction of the crash behavior of SFRP structures with very good accordance to physical tests. However, in early design steps often isotropic simulation approaches are deployed in order to save calculation time and license costs for additional software tools. But these isotropic approaches deliver results of just minor accuracy and therefore, do not support the product developer adequately at determining an optimum design in the early phases.

From the point of view of engineering design a compromise between highly accurate and cheap respectively quick simulation approaches has to be found. A simulation method for the early design steps should fulfill the following requirements:

- Delivering accurate results while considering all relevant effects of material behavior (anisotropy, strain-dependency, etc.)
- Enabling simulations with low computational and manual effort
- Ideally using standard CAE-software
- Providing reproducible results regardless of the CAE engineer's experience

A promising approach for early design steps implemented for LS-DYNA is presented by NUTINI in [5]. Mat_103 - which originally is designed to model the behavior of sheet metal structures - is used to represent the anisotropic material behavior of SFRP-structures. The determination of the material parameters is carried out automated by a multi-objective optimization. The mapping of the orientation is performed with the help of a self-developed mapping software, enabling reproducible results. The legitimacy of the approach could be proved by several experiments. However, the method is subject to certain restrictions. MAT_103 only allows the representation of anisotropic properties in the plastic phase and anisotropic failure criteria are not supported. The introduced mapping software is limited to a shell-mesh based process simulations. This only allows for a two dimensional flow analysis and consequently leads to minor accuracy. The consideration of the varying ODP in order to perform a local adaption of the stiffness is not mentioned.

Another interesting approach is introduced by SCHÖPFER [6]. Hereby, the complex material behavior of SFRP structures is modeled by *overlapping* two separate anisotropic material models (MAT_54 and MAT_108). MAT_54 covers the anisotropic failure behavior and MAT_108 enables the representation of anisotropic, non-linear stiffness properties. The superposition is realized layer wise within shell elements. To be precise, different material models are assigned to the integration points through the thickness of a finite shell definition with help of the *PART_COMPOSITE keyword. The introduced method is a phenomenological approach,

meaning that the orientation of the created layers is not equal to the layers appearing in the physical part as shown in fig. 1a. The method of overlapping material models can be considered very powerful, since almost arbitrary material behavior can be represented with the use of already implemented material models. The automatic extraction of the orientation data determined by an injection molding simulation was not the focus of the work presented in [6] – an interpretation of the process simulation “by hand” is proposed. Consequently, the repeat accuracy as well as the accuracy in general depends on the experience of the CAE-engineer. Nevertheless, tests described in [6] prove a very promising agreement between experimental and numerical results.

4. Phenomenological, integrative simulation approach of the KTmfk

In the following an approach of the chair of engineering design (KTmfk) enabling the simulation of SFRP-structures in early design steps will be described. The approach is based on the idea of SCHÖPFER [6], whereas the anisotropic MAT_108 is replaced by the isotropic MAT_98. On the one hand, this replacement leads to a reduced amount of input parameters and consequently reduces the complexity of the necessary parameter fitting process. On the other hand, strain-rate dependency can be covered with the use of MAT_98. In contrary to [6] an automated method extracting the orientation data of the injection molding is presented.

As already mentioned an integrative simulation approach for SFRP-structures needs a valid material model as well as a method allowing the consideration of the orientation data. The methods used for both tasks are discussed in the following two sections.

4.1 Applied material description

The material behavior of SFRP-parts is represented by overlapping the material models Mat_54 and MAT_98 which are characterized by a low amount of input parameters. Their calibration is performed within a reverse-engineering process. The material behavior determined in experimental characterization tests (tensile, shear and bending tests) has to be approximated by conforming virtual tests. The corresponding workflow is displayed in fig. 2 exemplarily for the tensile test.

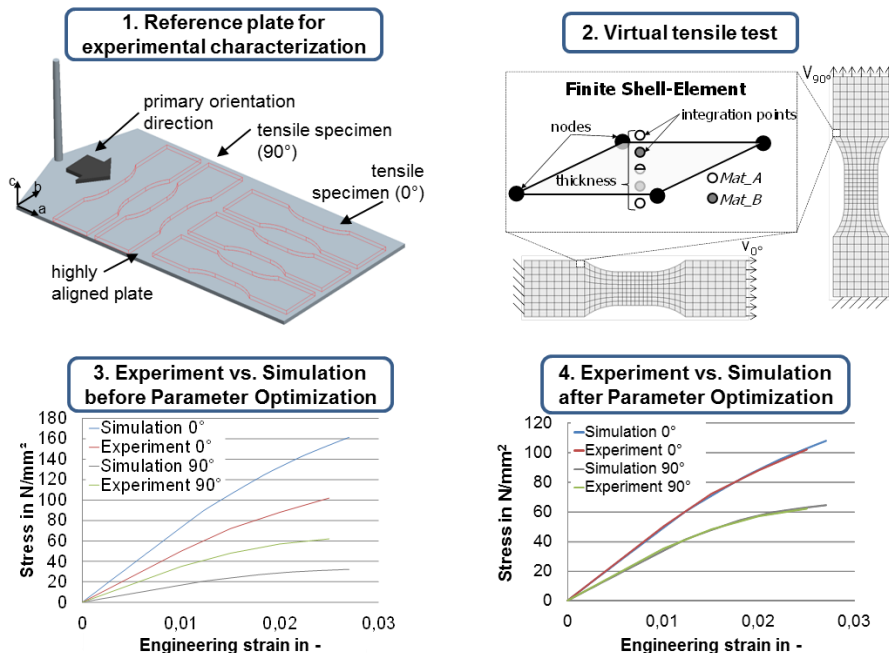


Figure 2: Fitting procedure for the tensile test

The test specimens for the experimental characterization have to be taken out of an ideally oriented reference plate. Since the anisotropic behavior is adjusted within the tensile fitting, specimens parallel (0°) and orthogonal (90°) to the major orientation direction are necessary. Within these tensile tests stress-strain curves of the focused polymer have to be collected. To be able to cover strain-rate dependency in the numerical model, the tests should be performed at different load velocities. Additionally, stress-strain curves have to be determined by Finite Element simulation with corresponding virtual tests. To be able to approximate the experimental material behavior with the numerical model the input parameters have to be adjusted adequately. The parameter fitting procedure for tensile, shear and bending tests is supported by using the optimization code LS-OPT[®]. Hereby, the bending stiffness is not controlled directly by material parameters, but by shifting the position of the layer and by adjusting the thickness of the layers of the shell description. Since the material models assigned to each shell element (see fig. 2.2) have different stiffness properties, placing stiff material models close to the surface leads to an increasing bending stiffness of the structure. Under the precondition that layers belonging to one finite element are oriented in the same direction, the final adjustment of the bending stiffness can be performed without affecting the already optimized tensile and shear behavior. Consequently, the layers assigned to each integration point of the shell elements do not represent the distinct layers of the physical model. That's why the presented approach has to be considered as phenomenological modeling method. A more detailed explanation of the material description and the parameter optimization can be withdrawn [7] respectively [8].

4.2 Consideration of orientation properties

The paper's focus is the transformation of the orientation data (orientation angle and orientation distribution probability) to the structural simulation. The procedure is divided in two steps, each of them supported by a software tool. In the first step (section 4.2.1) the orientation data obtained by the process simulation are averaged and assigned to the appropriate element IDs of the structural simulation. The adapted orientation data are collected within a text file. In the second step (section 4.2.2) an executable LS-DYNA input deck is created containing the orientation information.

4.2.1. Mapping and adapting of the orientation data

The workflow of the mapping and averaging process is displayed in fig. 3. First, the orientation data calculated at the nodes of the mesh of the process simulation are assigned to the corresponding element of the structural simulation. Therefore, a bounding box around the shells containing the corresponding nodes of the mesh of the process simulation is created. The orientation tensors assigned to each shell element are averaged component wise, delivering one single orientation tensor per element.

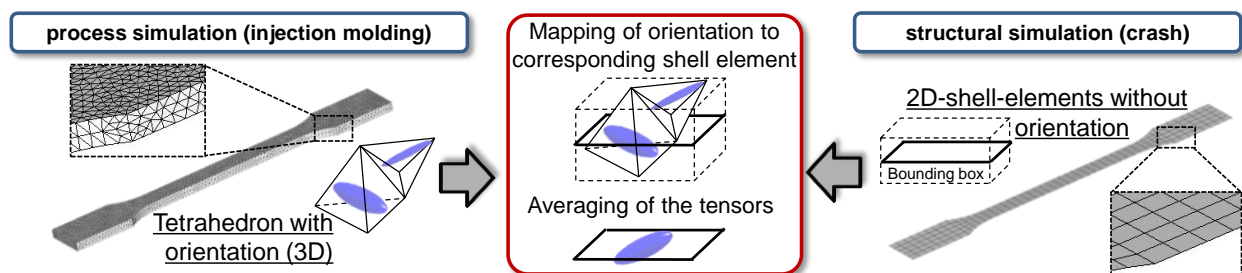


Figure 3: Workflow mapping process (to enable a clear arrangement, the orientation ellipsoids are displayed in the center of the tetrahedrons rather than at the nodes)

After the mapping process, the orientation data have to be adapted for the structural simulation. The anisotropic behavior of the material description introduced in section 4.1 is implemented with the use of MAT_54. The anisotropic stiffness in MAT_54 is defined by *two orthogonal young modul* values. Since shell elements are used, only stiffness properties in the shell plane can be considered. Consequently, the averaged orientation ellipsoid has to be projected on the shell plane; thereby, the stiffness share normal to the shell plane is neglected. Afterwards the x_a and x_b axis – the local element coordinate system – necessary to set the two young-moduli of MAT_54 have to be defined as displayed in fig. 4. Therefore, the three axis of the averaged orientation ellipsoid are projected on the shell's surface. The major orientation vector after the projection ($\lambda_1 e_1^*$ in fig. 4) defines the mayor axis x_a of the element coordinate system. The minor axis x_b of the element coordinate system can be derived from x_a , since x_b is orthogonal to x_a . After defining all element coordinate systems, the orientation data of each element $\lambda_i e_i$ have to be projected on the axis x_a respectively x_b and being normalized in the end.

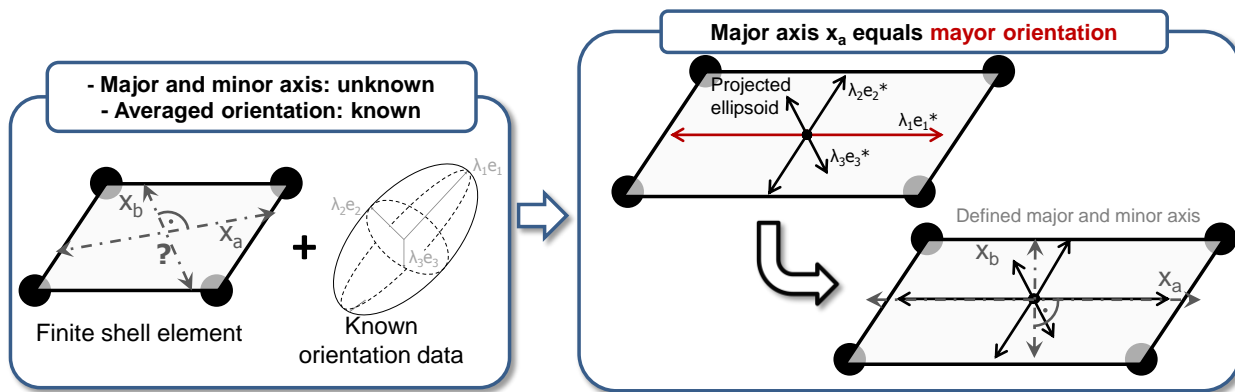


Figure 4: Determination of the alignment of the local coordinate system x_a/ x_b necessary for Mat_54

With help of this method, the appropriate alignment of the local coordinate system of each shell element as well as the corresponding ODP for the x_a and the x_b axis are determined. These steps are carried out automated with the software prototype called *KTmfk Mapping Tool*. By using the *KTmfk Mapping Tool* the orientation data from the process simulation are transferred to the mesh of the structural simulation without noticeable loss of information (see fig. 5).

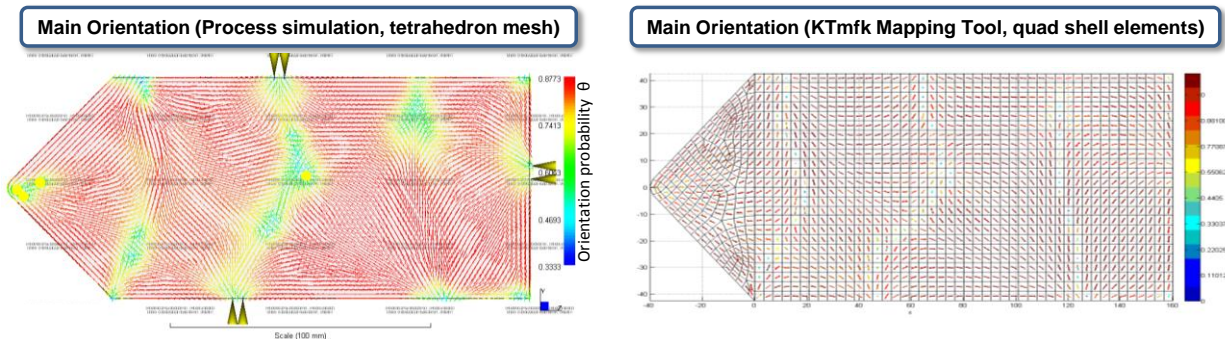


Figure 5. Injection molded plate with chaotic gate setup: Orientation data of process simulation vs. mapped data

However, the method described in section 4.2.1 so far does not provide any information regarding an alternating bending stiffness of the part, resulting from the fiber distribution across the thickness. Therefore, the bounding boxes surrounding each shell element (see fig. 3) have to be split horizontally in three sub boxes, allowing the determination of a layer wise orientation

information (see fig. 6). These data can be used for an additional adjustment of the bending behavior of the structural model.

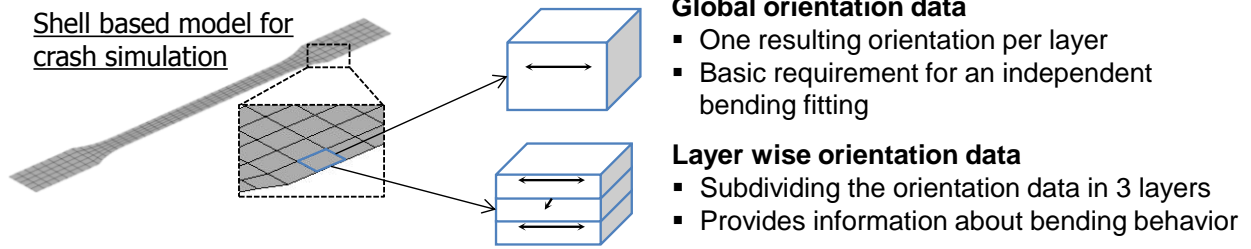


Figure 6: Different representations of the orientation data

4.2.2. Creation of an executable LS-DYNA input file

The methodology to create the input deck is embedded within a software prototype called the *KTmfk Input Deck Creator*. The pursued methodology can be explained most easily by demonstrating the workflow of the software. The user interface of the software is displayed in fig. 7.

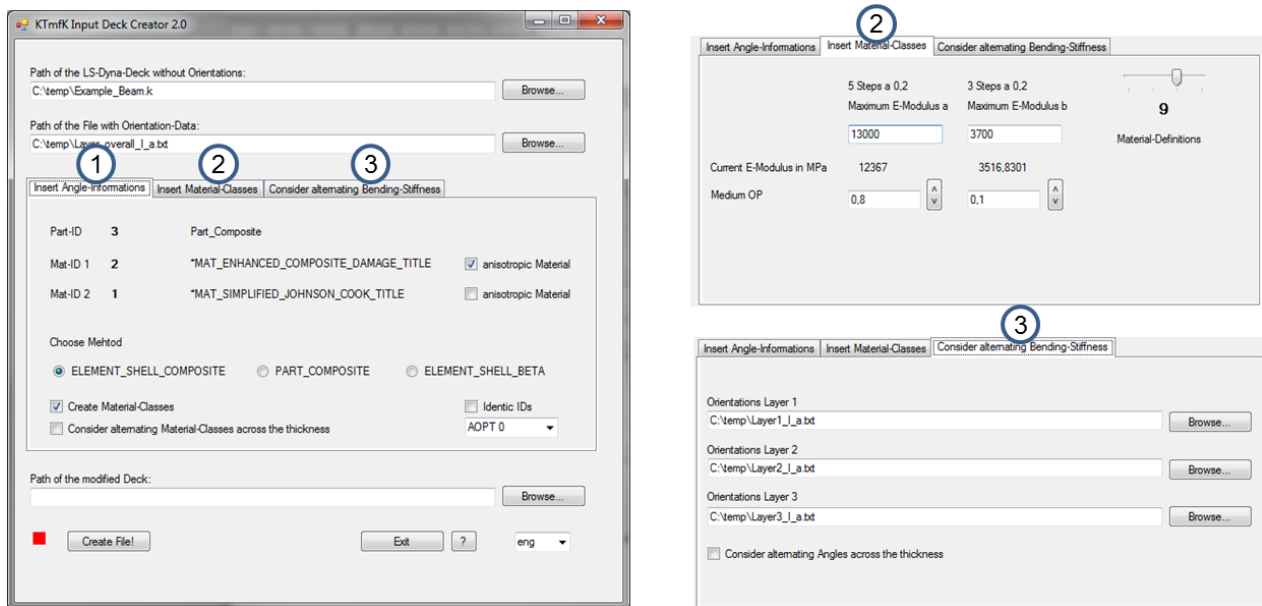


Figure 7: User interface of the KTmfk Input Deck Creator

In the beginning the simulation model of the laminar part to be analyzed is set up as .k-file using an arbitrary pre-processor. The part has to be represented by shell elements using the conventional *ELEMENT_SHELL definition. Within the model setup one *PART_COMPOSITE definition has to be declared. Two material models have to be assigned alternating to the integration points across the thickness of the SFRP-part. In the current version the combination of MAT_54 und MAT_98 are recommended, as explained in section 4.1. Furthermore, the path to the text file containing the orientation data, (visualized on the right side of fig. 5) created by the KTmfk mapping tool, has to be declared.

Tab 1: Insert Angle-Information

- Within this tab the desired element card being used to define the element representation including the orientation information has to be chosen. Hereby, the user can decide between a model setup using *ELEMENT_SHELL_BETA, *PART_COMPOSITE or the new *ELEMENT_SHELL_COMPOSITE keyword. The pros and cons of these three methods are discussed at the end of this section.
- Since the orientation angle of each element should be linked directly to each element (meaning in case of rotation of an element, the orientation angle should stick to the material), the orientation angle has to be defined using a local coordinate system, preferably by using the option Aopt=0. By doing so, even in case of large distortions or rotations of the element, a physically correct adjustment of the orientation angle can be guaranteed.
- The varying ODP leads to a locally alternating stiffness behavior. This phenomenon can be considered by activating the check box “Create Material classes”. This option only works in combination with the *PART_COMPOSITE or the *ELEMENT_SHELL_COMPOSITE definition.

Tab 2: Insert Material-Classes

- The different material classes being necessary to consider the varying ODP can be controlled within Tab 2. The material parameters of MAT_54 and MAT_98 determined by reverse-engineering are only valid for areas of the part which are characterized by an ODP state close to the mean ODP of the ideally oriented reference plate mentioned in section 4.1. The reference plate used for the current studies shows a mean ODP of approximately 0.8 parallel ($\Theta_a=0.8$) respectively 0.1 orthogonal ($\Theta_b=0.1$) to the major orientation direction as defined in fig. 2. The ODP share of 0.1 missing to sum up to 1 arises from fibers aligned normal to the surface of the part. Consequently, the stiffness behavior of areas of the part being characterized by an ODP deviating from $\Theta_a=0.8$ and $\Theta_b=0.1$ has to be increased respectively decreased adequately.
- Therefore, the ODP in a- and b-direction has to be divided in several classes. Table 1 shows a classification of nine different classes. The material parameters determined within the reverse-engineering are only valid for representing the ODP-state defined by class 4. For the remaining classes the stiffness has to be adjusted by alternating definitions of MAT_54, being defined by different values for the young moduli. These values are obtained by linear interpolation. MAT_98 which defines the curvature of the resulting stress-strain curve of the virtual tensile test is kept constant.

Table 1: Material classes for distinct sets of ODP-groups

		Θ_a				
		0.0 - 0.2	0.2 - 0.4	0.4 - 0.6	0.6 - 0.8	0.8 - 0.4
Θ_b	0.0 - 0.2	1	2	3	<u>4</u>	5
	0.2 - 0.4	-	6	7	8	-
	0.4 - 0.6	-	-	9	-	-

- To perform the interpolation, the young modul values for class 5 of table 1 have to be declared. The appropriate values were determined by preliminary tests. The amount of different material classes and consequently the level of detail of the simulation can be chosen within the software on tab 2.

Tab 3: Consider alternating Bending-Stiffness

- Finally an alternating bending stiffness resulting from the manufacturing process can be taken into account by assigning different material classes (and orientation angles) across the thickness of each element according to the results of the layer wise analysis of *KTmjk Mapping Tool* (see fig. 6). Therefore, the orientation information of three horizontally arranged layers have to be declared within tab 3. The positive influence of additionally assigning the alternating orientation angles of each layer has to be investigated in further studies.

Finally the three different element cards enabling the definition of the element representation will be discussed briefly. `*ELEMENT_SHELL_BETA` allows a compact element definition and a user-defined variation of the alignment of each element, defined by one orientation angle per element. Though, the adjustment of the bending stiffness by assigning different material classes according to the results of the fiber orientation distribution is not possible. The calculation time with `*ELEMENT_SHELL_BETA` is similar to the calculation with a standard `*ELEMENT_SHELL` definition (without an orientation declaration). The use of the `*PART_COMPOSITE` command enables the assignment of user-defined element angles *and* alternating material classes for each element and layer of the focused part. However, this flexibility requires the declaration of a `*PART` definition for each element and consequently leads to an extensive model definition. Besides the confusing structure of the resulting input deck, a noticeable increase of the calculation time is resulting. Based on the example of a virtual tensile test consisting of 105 shell elements the calculation time decreased by the factor of seven compared to a simulation without an orientation declaration. The new keyword `*ELEMENT_SHELL_COMPOSITE` can be seen as a compromise between the two previously mentioned keywords. On the one hand, it offers the flexibility within the model definition as provided by the `*PART_COMPOSITE` command. On the other hand, the input deck can be kept compact and clearly arranged. Although an increasing calculation time is observed, `*ELEMENT_SHELL_COMPOSITE` can be considered more efficient than `*PART_COMPOSITE`. Compared to a simulation without any orientation data, the elapsed time increases six-fold with the use of `*ELEMENT_SHELL_COMPOSITE`.

5. Definition of the failure criterion

The failure criterion for the introduced material description is implemented by different approaches (anisotropic and isotropic failure criterion).

Anisotropic failure criterion

Since the initial analysis of the failure criterion implemented in `MAT_54` did not deliver satisfactory results, an approach with two `*MAT_ADD_ERROSION` definitions assigned to `MAT_54` was pursued. Hereby, the alternating stress-strain behavior in longitudinal and transversal direction is taken advantage of. The maximum stress values can be observed at load paths parallel to the major fiber alignment (0° orientation). On the contrary the maximum strain level can be observed orthogonal to the major fiber alignment (90° orientation). Consequently, one `*MAT_ADD_ERROSION` limits the maximum permitted stresses (failure criterion 0°) the other `*MAT_ADD_ERROSION` definition limits the maximum strain (failure criterion 90°). Within tensile tests in 0° and 90° orientation a good correlation between simulation und experiments could be obtained (see fig. 8). For these studies experimental stress-strain curves published in [9] are taken.

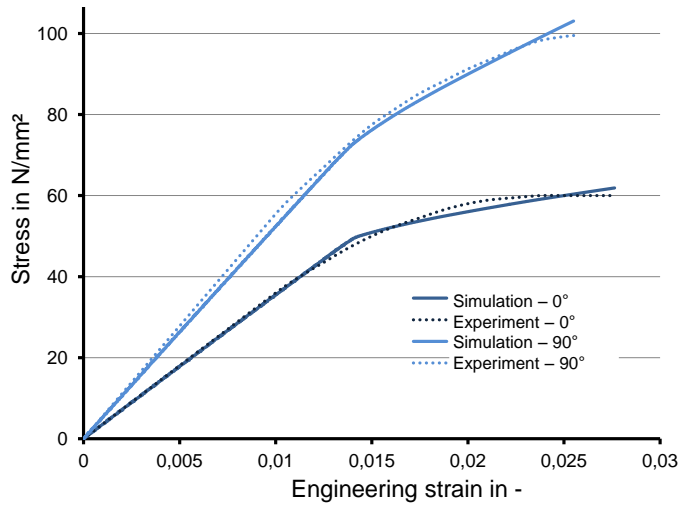


Figure 8: Representation of anisotropic failure within tensile tests

Isotropic failure criterion

The effective plastic strain will be considered as isotropic failure criterion. Within MAT_98 the effective plastic strain can be limited without an additional *MAT_ADD_ERROSION definition.

6. Validation

The Validation of the approach was performed in two steps. Initially, the energy ratio of a part with mixed material models was analyzed within a preliminary test. Finally, an experiment of a drop weight tester is modeled with the help of simulation. The obtained results of simulation and experiment were opposed and analyzed.

6.1 Preliminary validation test

The first preliminary test should prove that the mixture of several material models does not lead to corrupt energy ratios. Therefore, a simple model of a plastic girder was set up. This girder was loaded with an accelerated mass as displayed in fig. 9. The simulation was executed two times, one with MAT_ELASTIC and another one with the mixture of MAT_54 & MAT_98 assigned to the plastic part. The energy ratio displayed on the right side of fig. 9 proves that the mixture of the material models does not compromise the energy balance.

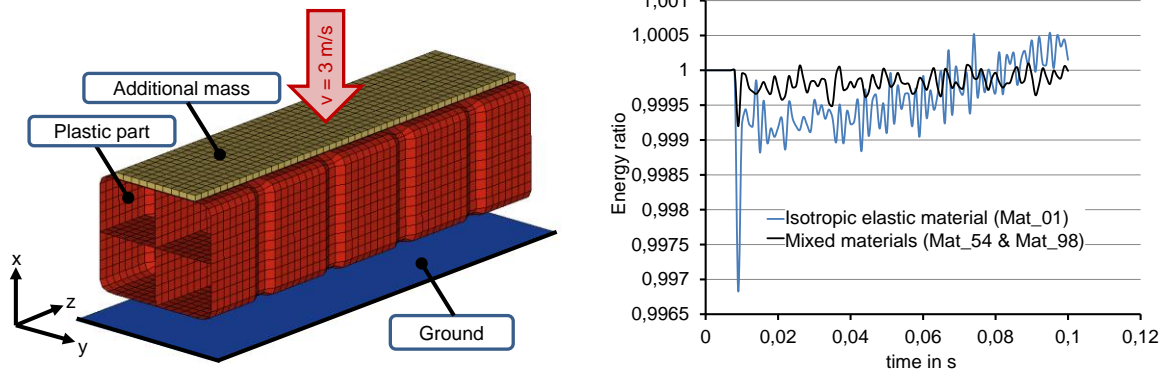


Figure 9: Simulation set-up and corresponding energy ratio check

6.2 Experimental validation of a virtual drop weight test

As experimental validation a drop weight test is recreated by simulation using the introduced material description. Within this test set-up a disc-shaped impactor hits a rectangular plastic plate (test object) at velocities of approximately 1.5 m/s respectively 2.1 m/s. The impactor and the guidance construction have a mass of 6.13 kg in total. This leads to impact energy values of 6.9 J respectively 13.5 J. The plastic plate is mounted on a clamping frame displayed in fig. 10.

As test object the reference plate mentioned in section 4.1 was used. These plates were manufactured by injection molding using the thermoplastic polymer Crastin[®] LW9020 NC010 from the manufacturer DuPont[™]. The plates show dimensions of 140 mm x 79.5 mm x 2.4 mm and are characterized by a homogenous orientation along the long edge of the plate (a-axis). This initial validation, for simplification purposes, is performed under the assumption of an ideal fiber orientation along the a-axis.

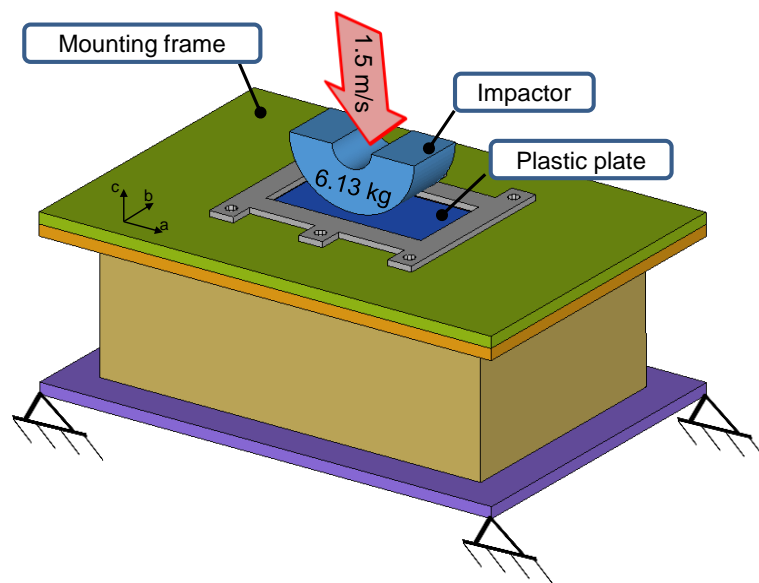


Figure 10: Test set-up of the drop weight experiment

Since the anisotropic failure criterion described in section 5 did not lead to satisfactory results for the component tests, the isotropic failure criterion limiting the effective plastic strain is deployed for the following validation tests. Pretests showed that an effective plastic strain threshold of 0.033 enables a good representation of the physical failure behavior.

The first test was performed with an impact velocity of 1.5 m/s. Two time steps of the experiment and the virtual model are displayed in fig. 11. The pictures of the first time step show an almost identical shape of the edge of rupture which is aligned parallel to axis of the major fiber alignment. The wider gap indicating the edge of rupture within the simulation model is a result of the deleted elements and therefore is dependent of the size of the mesh. The depth of the penetration apparent in time step two of the simulation is comparable to the experiment as well.

The drop weigh test was also performed with an increased impact speed of 2.1 m/s. The fracture pattern for this test is displayed in fig 12. Comparable to the results of the previous test a good correlation between physical and numerical test can be observed. The validation could prove that the presented simplified approach enables a realistic prediction of the crash behavior.

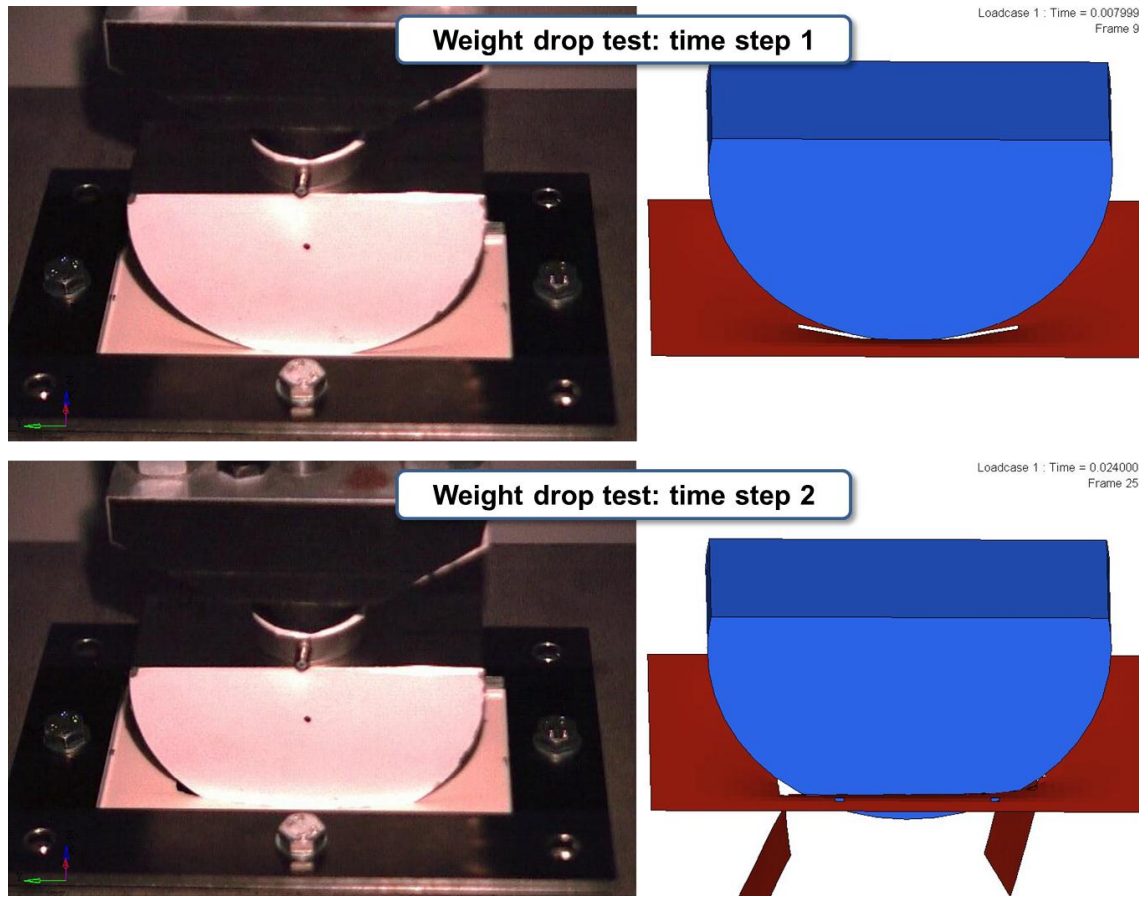


Figure 11: Comparison of physical and virtual test, impact speed: 1.5 m/s

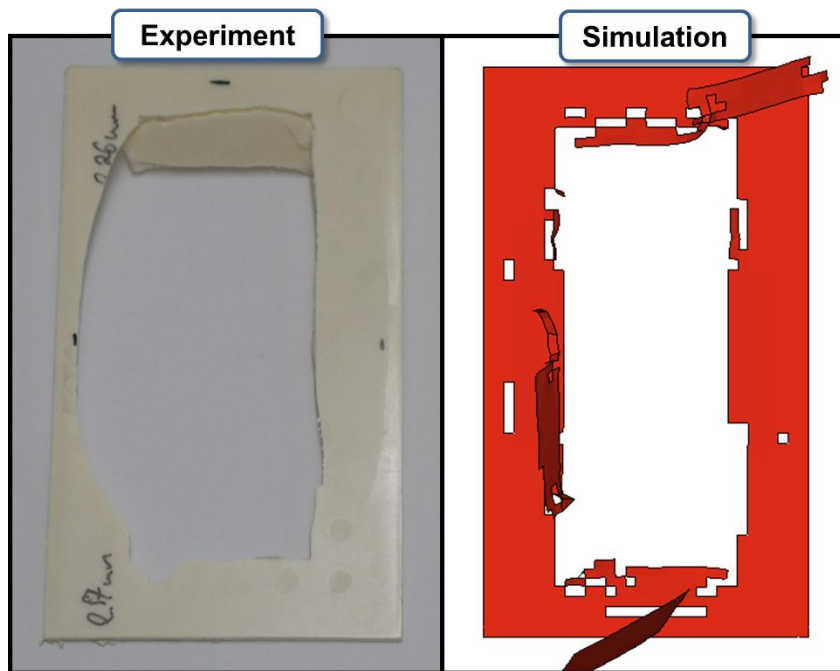


Figure 12: Fracture pattern of physical and virtual test, impact speed: 2.1 m/s

7. Summary and discussion

The present paper dealt with the prediction of the crash behavior of parts made of short fiber reinforced polymers. To enable an accurate structural simulation in early phases of the product development process, a simplified approach allowing an anisotropic crash simulation based on the orientation information obtained by an injection molding simulation was introduced. The complex behavior of short fiber reinforced polymers is represented by overlapping already implemented material models. Consequently, there is no need to implement any additional user-defined materials.

The orientation information delivered by the process simulation has to be prepared adequately to be inserted in the LS-DYNA input deck. Two software tools which support the CAE-engineer at preparing these orientation data and setting up the executable input deck were presented within the paper. Due to its flexibility and its justifiable calculation time, the deployment of the new *ELEMENT_SHELL_COMPOSITE keyword is recommended. The paper concludes with the description of an experimental validation, which proves the legitimacy of the approach.

The presented approach can be considered as suitable for early design steps, since standard CAE-software can be used. By using the technique of overlapping already implemented material models, even the complex material behavior of SFRPs can be modeled satisfactorily. Consequently, there is no need to handle the material description during the simulation by an external application, which is a common procedure in the field of integrative simulation. The deployment of the approach requires no additional manual effort, since the creation of the executable LS-DYNA input deck is supported by two software tools. So the simulation results do not depend on the CAE-engineer's experience. By supporting the use of the new *ELEMENT_SHELL_COMPOSITE keyword an acceptable calculation time for an anisotropic simulation can be achieved.

References

- [1] Advani, S.G.; Tucker III, C.L.: The use of tensors to describe and predict fiber orientation in short fiber composites. In: *Journal of Rheology* 31 (8), 1987, pp. 751–784.
- [2] Mlekusch, B.: Fibre orientation in short-fibre-reinforced thermoplastics II. Quantitative measurements by image analysis. In: *Composites Science and Technology* 59 (4), 1999, pp. 547–560.
- [3] Glaser, S.; Wüst, A.: Integrative crash simulation of composite structures. In: *5th European LS-DYNA Users Conference*, Birmingham, UK, 2005.
- [4] Seyfarth, J.; Hörmann, M.; Assaker, R.; Kattamuri, C.; Grass, B.: Taking into Account Glass Fibre Reinforcement in Polymer Materials: the Non Linear Description of Anisotropic Composites via the DIGIMAT to LS-DYNA Interface. In: *7th European LS-DYNA Users Conference*, Salzburg, Austria, 2009.
- [5] Nutini, M.; Vitali, M.: Simulating Anisotropy with LS-DYNA in Glass-Reinforced, Polypropylene-Based Components. In: *9th LS-Dyna Forum*, Bamberg, Germany, 2010.
- [6] Schöpfer, J.; Becker, F.; Maier, M.; Kolling, S.: Charakterisierung und Modellierung von kurzfaserverstärkten Kunststoffen - Teil 2: Simulationsmethoden mit LS-DYNA. In: *9th LS-Dyna Forum*, Bamberg, Germany, 2010.
- [7] Gruber, G.; Klein, D.; Wartzack, S.: A modified approach for simulating complex compound structures within early design steps. In: *8th European LS-DYNA Users Conference*, Strasbourg, France, 2011.
- [8] Klein, D.; Gruber, G.; Wartzack, S.: Materialparameter-Optimierung für die Crashberechnung in frühen Entwicklungsphasen. In: *Proceedings of the NAFEMS DACH-Conference*, Bamberg, Germany, 2012.
- [9] Krivachy, R.: Charakterisierung und Modellierung kurzfaserverstärkter thermoplastischer Kunststoffe zur numerischen Simulation von Crashvorgängen. Dissertation, Fraunhofer Institute for High-Speed Dynamics, 2004.

