THREE-POINT BENDING ANALYSES OF SHORT FIBER REINFORCED THERMOPLASTICS: A COMPARISON BETWEEN SIMULATION AND TEST RESULTS

***G. Gruber[1], S. Wartzack[1]**

[1]Chair of Engineering Design KTmfk, University of Erlangen-Nuremberg FAU, Germany

*Contact Author e-mail: gruber@mfk.fau.de

Abstract

In the context of lightweight design short fiber reinforced thermoplastics (SFRT) became increasingly popular due to their beneficial stiffness to weight ratio as well as their low fabrication costs. However, the prediction of the mechanical behaviour of parts made of SFRT by simulation is complex, because of the process induced fiber orientation. Especially in early design steps commonly imprecise isotropic simulation approaches are deployed in order to save time. In the present paper several anisotropic simulation approaches are evaluated regarding their modelling effort and accuracy.

*The paper starts with a description of the fundamentals of the anisotropic simulation of SFRT structures. Basically, the anisotropic properties of the part to be analysed are determined by an injection moulding simulation, which is linked with a structural mechanical simulation. Additionally, an anisotropic simulation methodology developed by the authors is introduced (IS4ED-approach – **I**ntegrative **S**imulation for **E**arly **D**esign Steps). Within three-point-bending analyses results delivered by the IS4ED-approach as well as by several commercial simulation tools are opposed experimental results. The analysed specimens are extracted from homogenous orientated plates 0°, 45° and 90° with respect to the preferred orientation. The accuracy of the focused simulation methods and the connected modelling effort are discussed.*

It can be noted, that all of the investigated simulation approaches can predict the general anisotropic behaviour of the test specimens quite well. However, the models created with the commercial tools are characterized consistently by a too stiff behaviour. The IS4ED-approach can predict the absolute displacement values more precisely. Furthermore, the FE-models created with the academic approach are significantly more compact, which makes their handling in the preprocessing more convenient.

Keywords: Short Fibre Reinforced Thermoplastics, Finite Element Simulation, Bending Analysis

Nomenclature

A_{ij}	2nd order fiber orientation tensor
A_{ijkl}	4th order fiber orientation tensor
E_{ii}	Young-Modulus in i-direction
G_{ij}	Shear-modulus in i-j-plane
L	Support distance of the 3-Point Bending Test
c_i	Transversally isotropic stiffness parameters
t	Thickness of test specimen
w	Width of test specimen
$\square_i$	Fiber orientation probability in i-direction
Φ^F	Fiber volume fraction
ν	Poisson's ratio

Abbreviations

CAE	Computer aided engineering
FEM	Finite element method
HT	Halpin-Tsai equations
MC	Material class
MF	Autodesk Moldflow Plastic Insight 2013®
SFRT	Short fiber reinforced thermoplastics
TW	Tandon-Weng equations

1. INTRODUCTION

The increasing scarcity of natural resources call for weight reduced and consequently more energy efficient solutions within almost all technical areas. Due to their beneficial stiffness to weight ratio as well as their low fabrication costs injection moulded, short fiber reinforced thermoplastics (SFRT) became increasingly popular [1] in the context of lightweight design.

Another key aspect to achieve substantial progress in lightweight design is the adequate exploitation of the early design steps, since they offer the maximum freedom of design. So on the one hand, the dimensioning of the part in early design steps should be performed with major accuracy to improve the lightweight quality. On the other hand, early phases call for time efficient methods. In general, the prediction of the mechanical behaviour of parts made of SFRT is complex, because of the process induced fiber orientation. To reduce the simulation effort, in early design steps frequently isotropic simulation approaches are deployed, even though these methods only allow for a poor prognosis quality and consequently often lead to overdesigned structures [2].

To achieve accurate results, the fiber orientation has to be taken into account. The orientation state can be determined by an injection moulding simulation. Methods combining a process simulation with a succeeding structural simulation are referred to as "integrative simulation" [3]. Initial approaches of this methodology are already implemented in commercial finite element applications.

Due to the importance of the early phases in the product development process, the following research question arises: How accurate are integrative simulation approaches and are they suitable for the early phases?

The present paper starts with a description of the fundamentals of the linear elastic integrative simulation and an academic integrative simulation approach developed by the authors. The accuracy of the academic approach and several commercial applications are evaluated within bending analyses. In conclusion, the simulation methodologies are discussed regarding their suitability for the application in early design steps.

2. PROCEDURE OF THE INTEGRATIVE SIMULATION

2.1 State-of-the-art in linear elastic modelling

Integrative simulation enables taking into account the fiber dependent stiffness and strength parameters including its spatial orientation. Since the focus of the current paper lies on stiffness analyses, the strength parameters will not be discussed in the following. An overview of the basic steps of the linear elastic integrative simulation is displayed in figure 1 and will be described in the following.

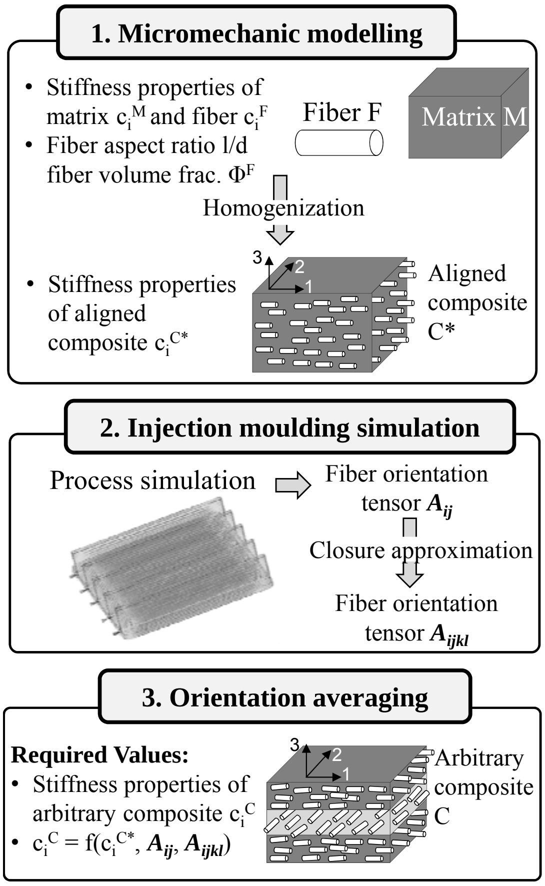

Fig. 1 Overview of the basic steps of the integrative simulation

Micromechanic modelling. In order to reduce the modelling effort within the structural simulation the heterogeneous matrix/fiber composite will be modelled as homogeneous continuum. In the context of homogenization, a transversally isotropic material behaviour is commonly assumed for the composite [4]. Consequently, the stiffness can be described by a set of five parameters: two young-moduli (E_{11} and E_{22}), two poisson's ratios (v_{12} and v_{23}) and a shear-modulus (G_{12}) [5]. With the help of homogenization techniques these five parameters describing the stiffness behaviour of the composite can be derived. The required input parameters are the isotropic stiffness parameters of fiber and matrix, the fiber volume fraction Φ^F and the fiber's aspect ratio (length/diameter). However, the stiffness properties (c_i^{C*}) delivered by common homogenization techniques are only valid for the assumption of a *perfect alignment condition* (see figure 1) [6]. Within the present paper the most widely used models [7] – Halpin-Tsai and Tandon-Weng – will be considered.

The semi-empirical Halpin-Tsai equations are a modification of Hermans model [8] for predicting the stiffness of continuous fiber composites. The analytical Tandon-Weng model is derived with the help of the mean-field theory, which enables the prediction of the global properties of a heterogeneous material. The sought stiffness properties are determined based on average stress respectively strain fields [9].

Injection moulding simulation

Besides the transversally isotropic stiffness parameters, information about the local orientation state is required to be able to set up an anisotropic finite element analysis. Therefore, local material coordinate systems have to be defined. This information can be determined by an injection moulding simulation. It delivers a 2nd order orientation tensor A_{ij} for each element of the mesh of the process simulation, describing the local orientation state. By performing a principal axis transformation, the principal orientation axis (eigenvectors) and the corresponding degree of orientation (eigenvalues) are defined. As shown in figure 2, the orientation state can be visualized as ellipsoid, with its principal axis coinciding with the eigenvectors. In case of a planar fiber orientation distribution the two eigenvectors coincide with the 1- and 2-axis of the transversally isotropic reference coordinate system.

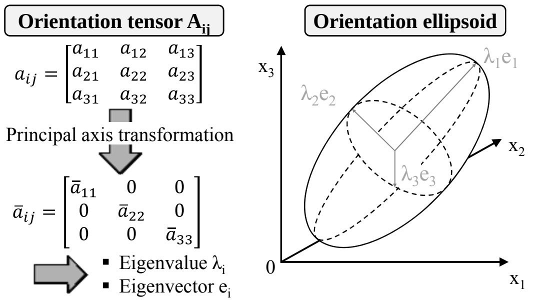

Fig. 2 Visualization of the orientation tensor

Orientation averaging

Since the stiffness properties determined by homogenization are only valid for a perfectly aligned orientation condition, a modification for arbitrary orientation has to be performed. The standard procedure is the so called orientation averaging by Advani/Tucker [10]. The required input values are the transversally isotropic stiffness parameters c_i^{C*} delivered by homogenization, the 2nd order orientation tensor A_{ij} as well as an additional 4th order orientation tensor A_{ijkl}. Since this 4th order tensor is not derived by injection moulding software, it has to be estimated by using a closure approximation. An overview of several closure approximations can be withdrawn Zheng et. al. [11]. The output of the orientation averaging are stiffness properties c_i^C for each finite element for the given orientation condition. With the help of the c_i^C the desired anisotropic structural simulation can be set up.

2.2 Integrative Simulation for Early Design Steps – IS4ED

Due to the importance of early design steps within the development process, the authors created a simulation approach (IS4ED) adapted to the needs of early phases. The goal was enabling an integrative simulation with reduced modelling respectively calculation effort and delivering accurate simulation results at the same time. The basic idea of the IS4ED-scheme is described in the following and summed up in figure 3.

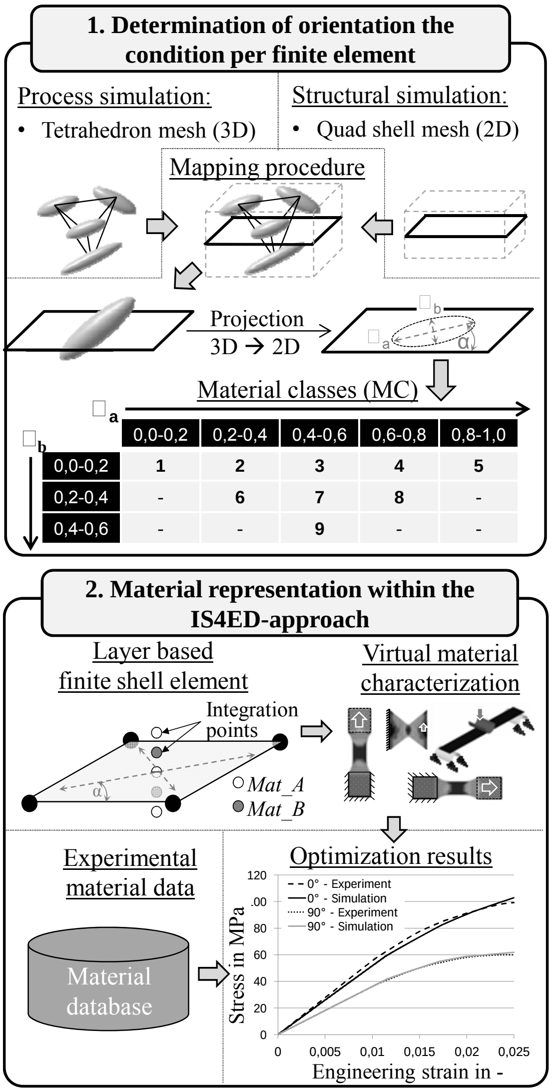

Fig. 3 Overview of the IS4ED-approach

To keep the calculation time of the anisotropic structural simulation on a reasonable level, the typically thin walled SFRP parts are modelled with a rather coarse shell mesh. In order to improve the quality of the results a quad mesh is used instead of a triangle mesh. The manufacturing dependent fiber orientation distribution is also determined via injection moulding simulation. However, for this process simulation a fine, three-dimensional tetrahedron mesh is deployed in order to take into account three-dimensional flow effects. Due to the mesh inconsistency between structural and process simulation a mapping methodology is proposed. Hereby, the orientation data of the process simulation are assigned to the corresponding quad shell elements. After performing an averaging procedure only one orientation tensor per quad element remains, describing its orientation condition. A detailed description of the deployed averaging method can be found in [12]. As a result of this procedure the orientation condition for each shell element of the structural simulation is simply described by three values. One reference angle α per element defines the orientation of the local transversally isotropic material description. Two further values describe the degree of the orientation ($\square_a$ and $\square_b$) in respect to the axis of the local material coordinate system (see section 1 of figure 3).

To extend the application area of the IS4ED-approach for additional fields like crash analysis, also effects like nonlinear behaviour and strain-rate dependency are taken into account. Since it’s difficult to cover all these effects within one material model, several material models are overlapped within one finite element description. To be more specific, the general anisotropic behaviour is covered by a linear elastic transversally isotropic material model (model b, see section 2 of figure 3). It is governed by the five material parameters described in section 2.1: two young-moduli (E_{11} and E_{22}), two poisson’s ratios (v_{12} and v_{23}) and one shear-modulus (G_{12}). The nonlinear stress-strain behaviour and the strain-rate/ temperature dependency (if required) are covered by adding an isotropic, elastic plastic material model (Johnson Cook Model [13]). It is defined by equation 1:

$$\sigma_y = \underbrace{\left(A+B\cdot\varepsilon_{pl}^n\right)}_{I}\underbrace{\left(1+C\cdot ln\frac{\dot{\varepsilon}_n^p}{\dot{\varepsilon}_0}\right)}_{II}\underbrace{\left(1-\left(\frac{T-T_r}{T_m-T_r}\right)^m\right)}_{III} \qquad (1)$$

Within the Johnson Cook model, the flow stress σ_y is a function of the yield strain ε_{pl} and the constants A, B and n (see term I). The strain-rate and the temperature dependency are taken into account by the terms II and III of equation 1. Since the analyses of this paper are carried out at constant temperature and are of static nature, these last two terms are set to 1 for the following considerations.

The mentioned numeric material parameters of both of the deployed material models are determined by reverse-engineering. In concrete terms, the results of virtual characterization tests are opposed corresponding experimental results. With the help of an optimization algorithm the numeric material parameters are adjusted in order to fit the experimental behaviour satisfactorily. The specimens treated in the experiments have to be taken from a reference plate characterized by a highly aligned orientation condition. Ideally, the experimental results are gathered from a material database.

However, the numeric material parameters determined by reverse-engineering, only represent the stiffness behaviour of local areas of the part, which are characterized by the same orientation condition ($\square_a$ to $\square_b$ ratio) characterizing the reference plate. In figure 3 the possible orientation states are divided exemplarily in 9 material classes, whereby the reference plate should by characterized by material class 5. Consequently, the material parameters determined by the optimization algorithm, are only valid for areas of the part which are described by material class 5. All other areas of the part – presumably the larger share of the part – are characterized by the remaining 8 material classes. These remaining classes are derived from the fitted material class (e.g. class 5). Hereby, alternating classes of the anisotropic material model (material model a) are created. The appropriate distribution of the distinct material parameters are estimated using the methodology of orientation averaging described in the previous section. The elastic plastic material model is kept constant, since the nonlinear stress-strain behaviour does hardly vary between the distinct material classes.

3. THREE-POINT BENDING ANALYSES

3.1 Test setup of the bending experiments

The three-point bending analyses are performed according to the standard DIN EN ISO 178 [14]. The specimens are loaded with a static force of 14.4 N. The dimensions of the test setup and the nominal dimensions of the specimens are displayed in figure 4.

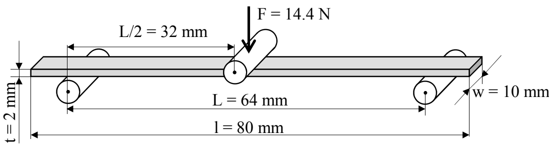

Fig. 4 Three point-bending test: DIN EN ISO 178

As displayed in figure 5, the test specimens are extracted from almost homogenous oriented plates at an angle of 0°, 45° and 90° with respect to the preferred fiber orientation by a milling process. As material a thermoplastic polyester resin (PBT+ASA) with 20% glass fiber reinforcement was chosen. Two samples of each specimen were extracted out of six plates (0° a S1 and 0° a S2, etc.). The exact dimensions of each individual specimen are listed in table 1.

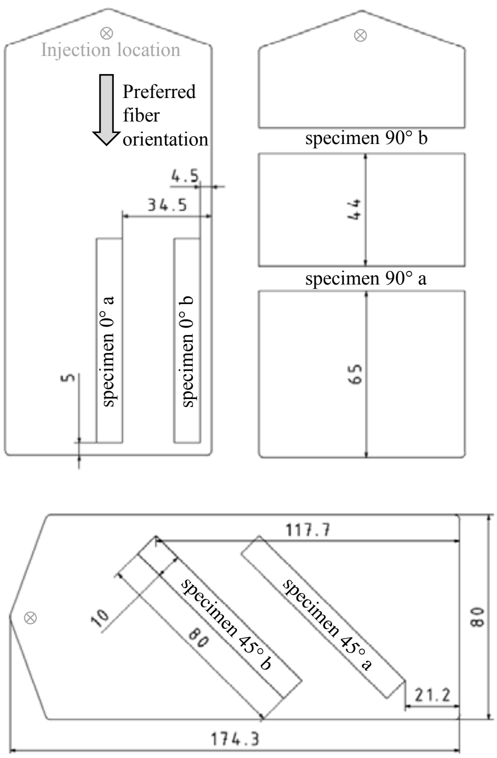

Fig. 5 Removal position of specimens within the reference plates

Specimen	w in mm (S_1/S_2)	t in mm (S_1/S_2)
0° a	10.17 / 10.38	2.35 / 2.30
0° b	10.38 / 10. 38	2.36 / 2.31
45° a	10.00 / 10.00	2.37 / 2.42
45° b	10.01 / 10.00	2.43 / 2.48
90° a	10.17 / 10.10	2.44 / 2.26
90° b	10.10 / 10.07	2.54 / 2.45

Table 1. Dimensions of the specimens

3.2 Model setup of the virtual bending tests

For all simulations performed in the present paper, the fiber orientation is derived using the commercial software Moldflow Insight 2013®. The manufacturing parameters in the simulation were chosen accordingly to the manufacturing conditions of the physical test plates mentioned in the previous subsection. The material definition of the focused polymer is embedded in the Moldflow material data base. The orientation condition is displayed in figure 6. The most accurate orientation condition can be observed parallel to the y-axis, close to the edge of the plate. By comparing figure 5 and 6, it can be stated that for each group of specimens (0°-, 45°- and 90°-specimens) the samples of removal position "b" are characterized by a slightly stronger orientation condition.

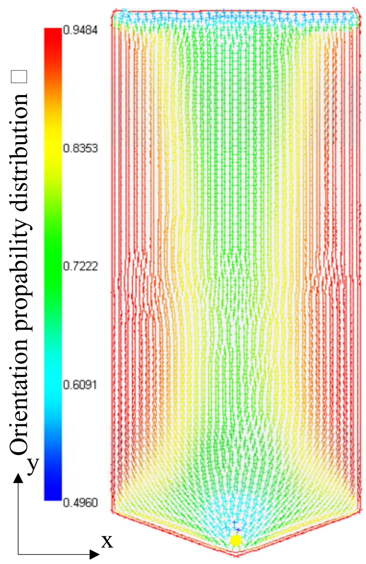

Fig. 6 Orientation condition of reference plate

With help of the CAE data interface of Moldflow executable input decks containing the anisotropic material properties can be exported. For the calculation of the stiffness properties the default orthotropic Closure Approximation of Moldflow was chosen. Within the following analyses the coupling with the solvers Ansys®, Radioss® and LS-Dyna® (implicit) was investigated, whereas the initial stresses delivered by Moldflow were neglected. Furthermore, the academic approach IS4ED – based on the solver LS-Dyna (implicit) [13] – was examined.

The investigated commercial CAE data interface requires a triangle shell mesh for the injection moulding simulation. For the succeeding structural simulation the same shell mesh is used. Since within the structural simulation only a subsection of the plate is analysed, the mesh has to be prepared regarding the shape and the location of the test specimens (see figure 7a).

Due to the mapping routine of the IS4ED-approach there are no restrictions regarding the meshes. The process simulation was setup using the preferred 3D tetrahedron mesh. For the structural simulation a rather coarse quad shell mesh was chosen. The dimensions of each virtual test specimen are chosen according to the dimensions of the corresponding physical test specimen (see table 1).

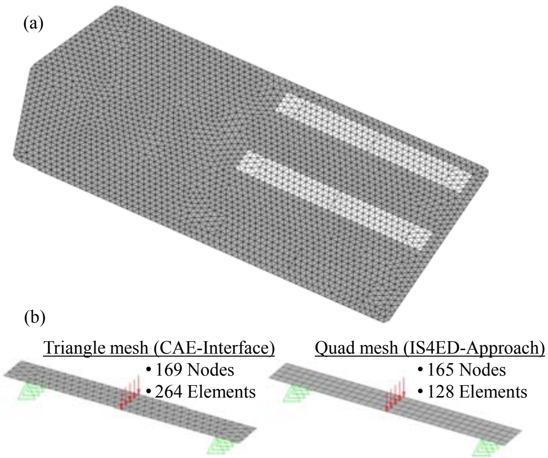

Fig. 7 a) Mesh of the plate used for the CAE data interface, b) Meshes of the structural simulations

To guarantee the comparability of both structural simulations, the meshes shown in figure 7b are characterized by the same order (1st order elements) and have almost the same number of nodes. Comparative simulations proved that if using an identical, isotropic material model both meshes deliver almost identical displacement results (deviation less than 0.1 %). Since displacement values delivered by both meshes are close to the analytical solution, both models can be considered as satisfactory and comparable.

In the following the structure of the models of the structural simulation will be described. Regardless of which finite element code is addressed by the CAE data interface, all resulting models have a similar structure. Each shell element is characterized by 20 layers respectively integration points through the thickness. All elements have a unique thickness distribution which is specified by one integration rule per element. For each layer of each element a unique material description is created. The degree of detail cannot be reduced within the settings of the commercial interface.

The IS4ED-scheme uses seven layers per element, with a constant thickness distribution for all elements. The amount of material-models to be used can be defined by the user and ranges from 1 – 30 anisotropic material models. A detailed comparison of the model structures is summed up in table 2.

	MF + Ansys/ LS-Dyna/ Radioss (CAE interface)	**IS4ED**
No. of elements	264 / 264 / 264	128
No. of integration rules	264 / 264 / 264	16
No. of material models	5280 / 5280 / 5280	1 - 30
Memory capacity	1.96 MB / 2.54 MB / 0.97 MB	0.06 MB

Table 2. Comparison of the model structure: Moldflow (MF) Interface and IS4ED

3.3 Results of the bending analysis

In figure 8, exemplarily the results of the simulation performed with Radioss of specimen “0° a S1” are displayed. The maximum displacement can be observed at the centre of the specimen (0,803 mm). Since the crucial difference between all the simulation results is the maximum displacement, only this characteristic will be analysed in the following by opposing the determined values within diagrams.

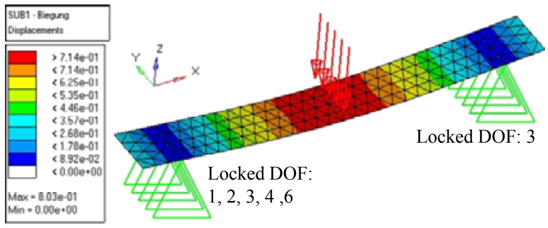

Fig. 8 Contour plot of displacement of specimen 0° a S1 (simulation performed with Radioss)

Figure 9a shows the maximum displacement *u* of the specimens. In this case the fibers are oriented parallel (0°) to the long edge of the test object. The stiffness properties utilized within the commercial integrative simulations are calculated using the Halpin-Tsai approach. For the first set of simulations, only one class of anisotropic material models (1 MC) is used, meaning that varying degrees of the orientation condition are neglected. Since, the focused reference plates are characterized by an almost constant orientation condition (see figure 6), this simplification can be considered as justifiable. Furthermore, this hypothesis is commonly assumed within simplified simulation approaches [15], [16].

As the material parameters of the IS4ED-approach are determined by reverse-engineering, no homogenization technique is deployed. The deviations between simulation and experiments (see figure 9b) show that the IS4ED-approach can predict the bending behaviour with the highest accuracy. The CAE data interface consistently models a too stiff behaviour. This effect cannot be traced back to the linear triangle mesh (see section 3.2), since the deviation of the displacement delivered with both meshes was about 0.1 %. The ratio of the displacements between the four test specimens can be predicted satisfactorily with each simulation method.

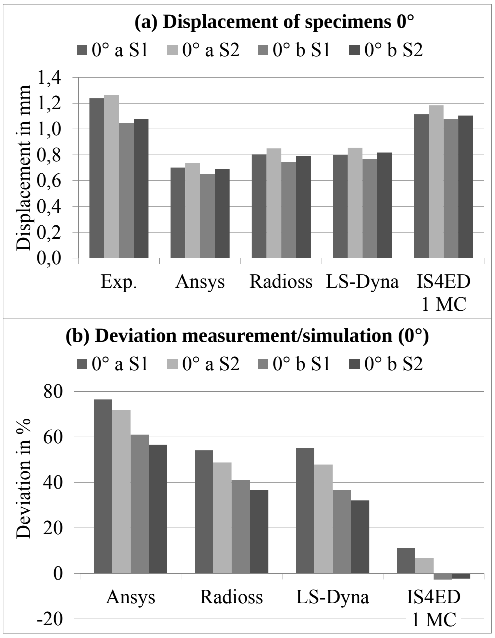

Fig. 9 Displacements of specimens 0° (CAE data interface using Halpin-Tsai)

Figure 10 shows the same results as discussed above, but for specimens with a fiber orientation of 90°. The findings that can be drawn from these test results are similar to the ones gained within the previous analysis.

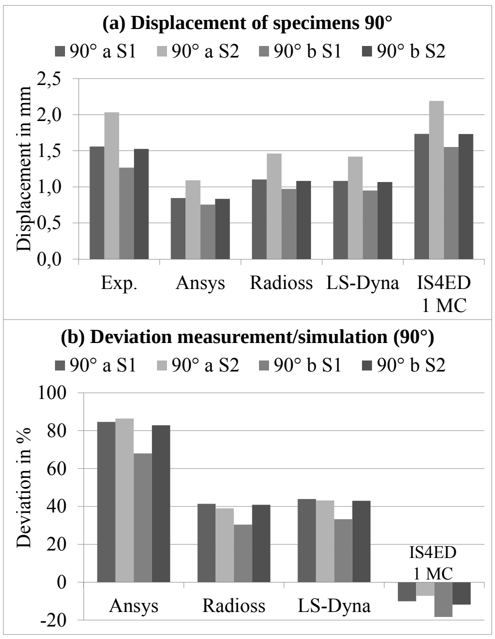

Fig. 10 Displacements of specimens 90° (CAE data interface using Halpin-Tsai)

The relative deviation of the stiffness of the specimens extracted at an angle of 45° can also be predicted with sufficient accuracy (see figure 11). It can be stated that the absolute deviation is smaller compared to the previous analyses. The most accurate solution is obtained again with the use of the IS4ED-approach.

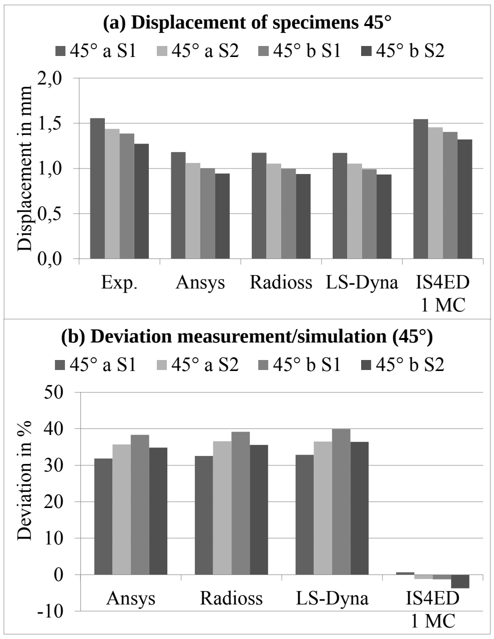

Fig. 11 Displacement of specimens 45° (CAE data interface using Halpin-Tsai)

For the commercial CAE interfaces, the influence of the most common homogenization techniques [Halpin-Tsai (HT) and Tandon-Weng (TW)] will be investigated as well. For an aspect ratio of 25 (≙ aspect ratio of the analysed polymer grade) the equations of Tandon-Weng lead to an increased stiffness in 0° (major E_{11}-value) and a minor stiffness in 90° (minor E_{22}-value) direction. As can be seen in figure 12, these presumable deviations of the stiffness are delivered by the simulation as well.

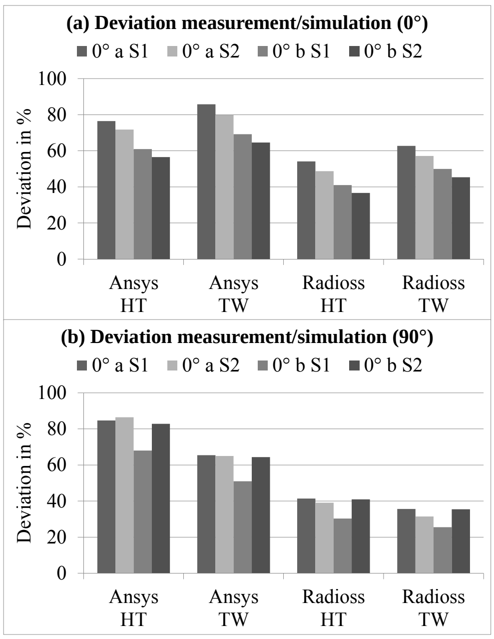

Fig. 12 Displacements of specimens 0° and 90° Halpin-Tsai vs. Tandon-Weng

Besides the fibre orientation condition, the bending stiffness of the specimens is also influenced by the varying dimensions of the specimens (see table 1) and the resulting section modulus. In the following, it will be investigated to what extent the bending stiffness is influenced primarily by the different states of fiber orientation. An appropriate characteristic describing the bending stiffness regardless of its geometry is the bending modulus E_B. According to [14] the bending modulus of a bending beam is defined as the quotient of the bending stress σ_b and the bending strain ε_b. The bending stress and the bending strain are defined as follows:

$$\text{(a)}\ \sigma_b=\frac{3FL}{2wt^2}\text{; (b)}\ \varepsilon_b=\frac{6ut}{L^2} \qquad (2)$$

In the previous equations u is defined as the maximum displacement. The further variables are described in figure 4. The quotient of σ_b and ε_b leads to

$$E_b=\frac{3FL^3}{12u\cdot wt^3} \qquad (3)$$

which defines the bending modulus E_b. Replacing the displacement u of equation 3 with the formula of the displacement of a isotropic three-point bending beam

$$u_{iso}=\frac{FL^3}{4wt^3E} \qquad (4)$$

leads to

$$E_b\approx E \qquad (5)$$

Equation 5 shows, that E_b corresponds to the young-modulus and consequently is a characteristic defining the stiffness independent from the geometry. However, since the alternating thickness of the test specimens lead to alternating fiber orientation conditions, E_b is not entirely independent from the dimensions of the specimens.

Due to the previous explanations, the following results can be expected: As the pairs of specimens extracted at position “b” are characterized by a slightly stronger orientation condition, these specimens should show a stronger bending modulus. The pairs of specimens S_1 and S_2 (0° a S1 and 0° a S2, etc.) are extracted each from the same position of the reference plate. Consequently, these pairs should be characterized by a comparable bending stiffness.

The analyses of the bending modulus are performed exemplarily for the 0°- and 45°-specimens. The corresponding results are displayed in figure 13. As expected, it can be observed that the specimens of removal position b are characterized by a stronger bending stiffness. The relative deviation of the bending stiffness determined within the experiment can be predicted approximately by simulation. However, it must be noted that the test results are also connected with uncertainties. Since the bending stiffness of the specimens of each removal position differ only slightly, the use of just one material class within the IS4ED-approach can lead to an imprecise prediction of the alternating bending stiffness. Figure 13 indicates that by using 30 material classes instead of one material class the prediction quality could be improved, especially in the case of the 0° test specimens.

The assumption, that specimens extracted at the same position are characterized by an identical bending stiffness can be verified by simulation, but not by the experiments.

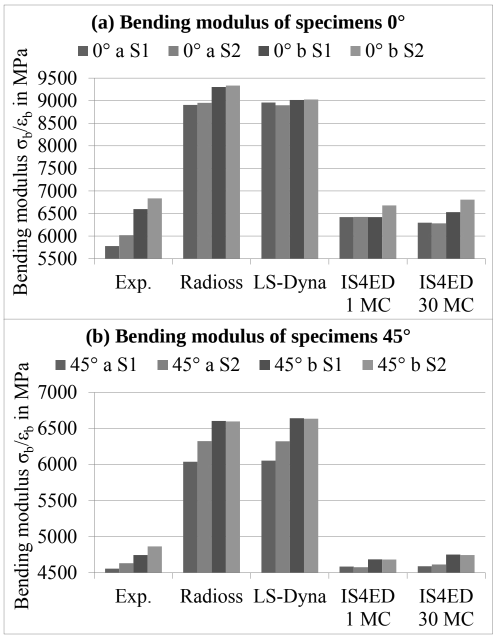

Fig. 13 Bending modulus E_b of test specimens 0° (a) and 45° (b)

3.4 Discussion

Finally, the CAE data interface as well as the IS4ED-apporach will be evaluated regarding their potential for the deployment in early design steps. In this context, it is important that the required input parameters for the simulation are easily accessible. Furthermore, the simulation scheme has to allow for a quick model creation and a short calculation time. In order to exploit the early phases adequately results with a sufficient accuracy have to be derived.

When using the commercial CAE data interface all required material data are included in the extensive material database of Moldflow. The required test results necessary for fitting the virtual material tests within the IS4ED-procedure were taken from the literature [17]. In case no appropriate material data can be found, time consuming characterization tests are inevitable.

The creation of the anisotropic part definition is performed automatically by all evaluated CAE tools. However, the extensive FE-models created with the CAE data interface complicate their handling within the preprocessing. To illustrate the size of the input decks created by the CAE data interface, the models of the bending beams are opposed to a full vehicle crash model. For example a FE crash model of a 2010 Toyota Yaris [18] with more than 1,000,000 elements contains approximately 770 material models in total. The investigated bending specimen of 264 elements contains 5280 material descriptions. These - element wise - small models can be treated within common preprocessing software quite well. Larger models (e.g. the plate shown in figure 5a with approx. 5000 elements and approx. 100,000 material models) frequently cause crashes of the preprocessing software.

By using the IS4ED-approach the absolute values of the displacement of the specimens can be predicted quite well. The numerical models created with the CAE data interface were characterized by a strong mean shift. The derived results were consistently too stiff. However, since the deviations can be considered as roughly constant an adjustment of the material data could improve the overall results of the CAE data interface significantly. Slight deviations of the bending stiffness (see figure 13a and b) could only be predicted qualitatively in the performed analyses. To improve the accuracy of the IS4ED-approach the deployment of several material classes is advisable.

Since the models created by the CAE data interface are linear elastic, in spite of the large input decks a minor calculation time can be guaranteed. Although, the models created by the IS4ED-approach contain non-linear material descriptions, the calculation time is also on a minor level, since the models are very compact.

4. SUMMARY

In the present paper several approaches for the Integrative Simulation of parts made of short fiber reinforced thermoplastics were evaluated based on stiffness analysis. Furthermore, the fundamentals of the investigated simulation methods were described and their potential regarding the suitability for early design steps was discussed.

Summing up, it can be noted that the commercial CAE data interfaces allow predicting the anisotropic character in general. Due to the strong deviation of the absolute displacement results, the material data (stiffness properties) provided by the injection moulding software has to be modified where required. As a result of the large input decks of the models, the analysed commercial method is restricted to parts with a minor amount of elements. The IS4ED-approach delivers the best results with models of reduced complexity. By increasing the amount of anisotropic material models from one to 30, the accuracy can be further improved. However, the necessary material parameters can only be found for several polymer grades in the literature. The development of a material database containing the required stress-strain curves for the reverse-engineering procedure is advisable.

The comparison of the models created with the CAE data interface with the models created with the IS4ED-approach shows that slightly alternating fiber orientation conditions are captured more accurately by the CAE interface. However, the compact structure of the input decks and major accuracy of the absolute results are reasons for using the IS4ED-approach in the context of early design steps.

5. REFERENCES

[1] Adam, H., *Carbon fibre in automotive applications*. Material & Design, Vol. 18, Issue 4, pp. 349-355, 1997.

[2] Satheesh Kumar, P., Srikari, S., Mahesh, N.S., Reddy, S., *Structural analysis of fiber-filled plastics with moulding process induced anisotropy*. SAS Tech Journal, Vol. 8, Issue 2, pp. 19-26, 2010.

[3] Michaeli, W., Baranowski, T., *Simulation of the Microstructure Formation in Injection Molded Thermoplastic Parts*, Journal of Polymer Engineering, Vol. 30, Issue. 1, pp. 29-42, 2010.

[4] Mlekusch, B.: *Thermoplastic properties of short-fibre-reinforced thermoplastics*. Composite Science Technology, Vol. 59, Issue 6, pp. 911-923, 1999.

[5] Stommel, M., Stojek, M., Korte, W., *FEM zur Berechnung von Kunststoff- und Elastomer-bauteilen.* Hanser Verlag, Germany, 2011.

[6] Tucker, C.L., Liang, E., *Stiffness Prediction for Unidirectional Short-Fiber Composites: Review and Evaluation*. Composite. Science. Technology, Vol. 59, Issue 5, pp. 655-671, 1999.

[7] Brandt, M., *CAE-Methoden für die verbesserte Auslegung thermoplastischer Spritzgussteile*. Ph.D. Thesis, RWTH Aachen, 2006.

[8] Hermans J., *The elastic properties of fiber reinforced materials when fibers are aligned*, Proceedings. Kon. Ned. Akad. V. Wetesch., pp. 1-9, 1967.

[9] Tandon, G. P.; Weng, G. J., *The effect of aspect ratio of inclusions on the elastic properties of unidirectionally aligned composites*. Polymer Composites, Vol. 5, Issue 4, pp. 327–333, 1984.

[10] Advani, S., Tucker, C.L., *The use of tensors to describe and predict fibre orientation in short fibre composites*. Journal of Rheology, Vol. 31, Issue 8, pp. 751-784, 1987.

[11] Zheng, R., Tanner, R.I., Fan, X.-J., *Injection Molding – Integration of Theory and Modeling Methods*, Springer Verlag, Germany, 2011.

[12] Gruber, G., Klein, D., Ziegler, P., Wartzack, S., *Consideration of Anisotropic Material Properties in Mechanical Design within Early Design Steps*, Proceedings of the 9th Norddesign, Aalborg, August, 2012.

[13] Livermore Software Technology Corporation, *LS-Dyna Keyword User's Manual*, V971 R 6.0, 2012.

[14] DIN EN ISO, *Plastics – Determination of flexural properties*, DIN EN ISO 178 Standard, 2011.

[15] Hauck, C., Brouwers, G.: *Faserverstärkte Spritzguss-Bauteile optimal auslegen.* Kunststoffe Vol. 82, Issue 7, pp. 586-590, 1992.

[16] Nutini, M., Vitali, M., *Simulating Anisotropy with LS-DYNA in Glass-Reinforced, Polypropylene-Based Components*, Proceedings of the 9th LS-Dyna Forum, Bamberg, 2010.

[17] Krivachy, R., Riedel, W., Weyer, S., Thoma, F., *Characterization and modeling of short fibre reinforced polymers for numerical simulation*. International Journal of Crashworthiness, Vol. 13, Issue 5, pp. 559-566, 2008.

[18] National Crash Analysis Center, http://www.ncac.gwu.edu/vml/models.html, Retrieved 20th Nov. 2012.