Considering the Local Anisotropy of Short Fibre Reinforced Plastics: Validation on Specimen and Component

R. Steinberger[1], T. Gross[1], P. Reithofer[2], St. Paul[3]

[1]Hirtenberger Automotive Safety GmbH & Co. KG
[2]4a engineering GmbH
[3]SimpaTec GmbH

1 Introduction

Finding the optimal design of plastic components using FEM tools has become an increasingly important topic for companies and research institutions. However, typical simulation models neglect the local anisotropy of injection moulded plastic components and often fail to predict the correct mechanical behaviour and failure mode. Using an integrative simulation approach, the manufacturing process and the resulting local anisotropies can be incorporated in a structural finite-element model which significantly enhances the predictive quality of the model. This is especially the case for glass-fibre reinforced polymers, where the difference between the mechanical behaviour longitudinal and lateral to the fibre is high. This also concerns the strength of weld lines, which are often inevitable in injection-moulded parts. Fibre orientation needs to be considered because it is indispensable for understanding weaknesses of a polymer part design.
Injection-moulding simulation is already a common tool for process simulation and allows calculating the fibre orientation. For structural analyses, various anisotropic material models have been developed or improved in the last years and interfaces to transfer the orientation become more and more available. This now allows making the step to integrative simulation also on an industrial level.
The anisotropic material model, used in this study is **`*MAT_4a_MICROMEC`**, a micromechanics model based on the Mori-Tanaka Meanfield Homogenization [1]. In this study, the experience of implementing an integrative simulation approach is presented. Furthermore, the fibre orientation determined by mould-flow simulation and the prediction of mechanical behaviour on specimen and component level are validated.

2 Material Characterisation

In the first phase, an anisotropic **`*MAT_4a_MICROMEC`** material model was calibrated for a 30% glass-fibre reinforced and impact-modified polyamide (PA6 GF30). Plates with the dimensions 80×80×4 mm³ were manufactured by injection moulding and tensile and bending specimens were machined at angles of 0°, 45° and 90° to the flow direction (Fig. 1 a). The specimens were conditioned according to ISO 1110 [2]. Tensile and bending tests were performed at different strain rates using a universal testing machine, a servo-hydraulic testing machine and a *4a-impetus* testing device. On the universal and servo-hydraulic testing machine, digital image correlation was used to capture the strains on the specimens. A subset of these data is shown in Fig.2.

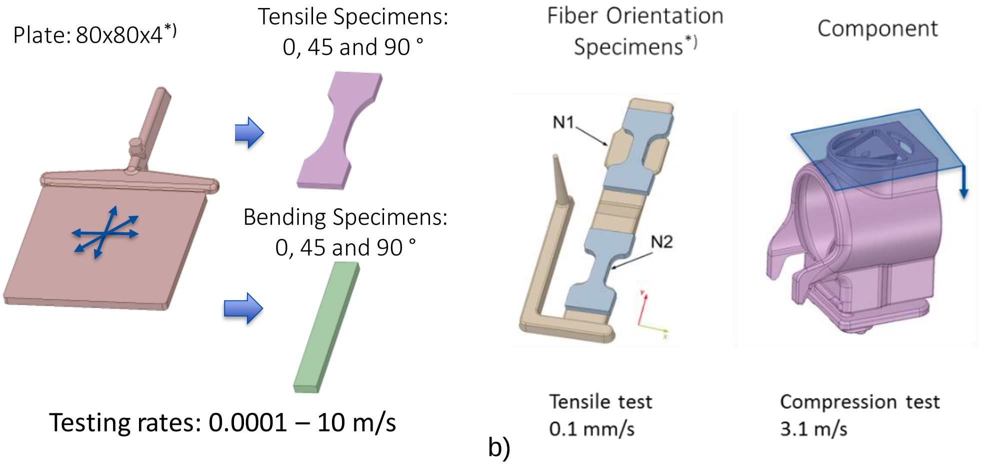

Fig.1: Specimens and Testing for a) material characterisation and b) validation.

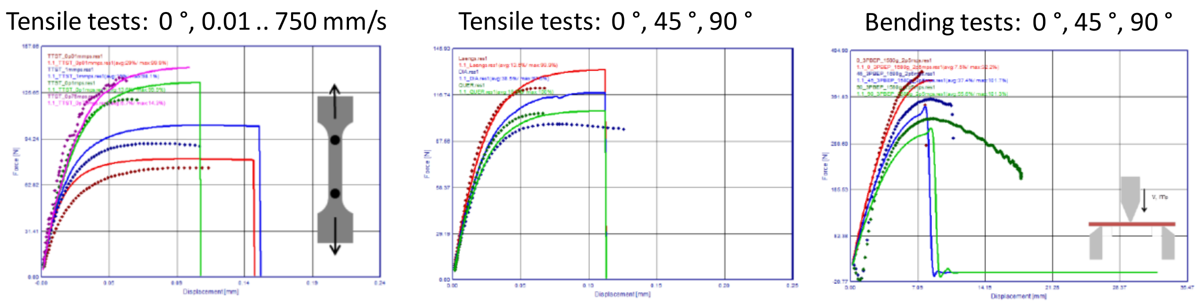

*Fig.2: Selection of test data for various load cases and loading rates with the simulation curves of the ***MAT_215** material model, which was fitted with 4a-fibermap software. The dotted curves show the force-displacement curves of the experimental test, the simulated curves are represented by the solid lines.*

The fibre-orientation distributions of these specimens were analysed by micro-computed tomography (µCT). These data were used to parameterise a material model by the reverse-engineering technique using the *4a-impetus* software. The main goal in the reverse-engineering process was to get a material card that can be used for dynamic load cases. Common material models used in engineering like *MAT_215 in LS-Dyna do not take viscoelasticity into account. Thus, a Young's modulus representing a higher loading rate was used to fit the test data. The resulting force-displacement curves are also shown in Fig.2. The fit describes the deformation behaviour reasonably well. In the case of the failure behaviour, the challenge was to find a compromise in matching the different load cases. This topic will be focus of further investigations.

3 Analysis on Fibre Orientation Specimens

In the second phase, the injection-moulding simulation and the material model were validated using tensile specimens with uniform geometry but varying fibre-orientation distributions over thickness; these are denoted here as fibre-orientation specimens. The fabrication of these specimens was simulated using the injection-moulding simulation software Moldex 3D R14. The simulations were compared with fibre-orientation distributions determined from µCT scans using the software tool VG Studio MAX 2.2. Theoretical background on the fibre-orientation tensor and the models used in injection moulding simulations can be found elsewhere [3].
The injection-moulding simulation could qualitatively represent the orientation distribution of the tensile samples. For sample N1, however, the degree of orientation in the shear layer of the testing plate was overestimated while the formation of a mid-layer in the core of the samples was quantitatively underestimated (Fig. 3a).

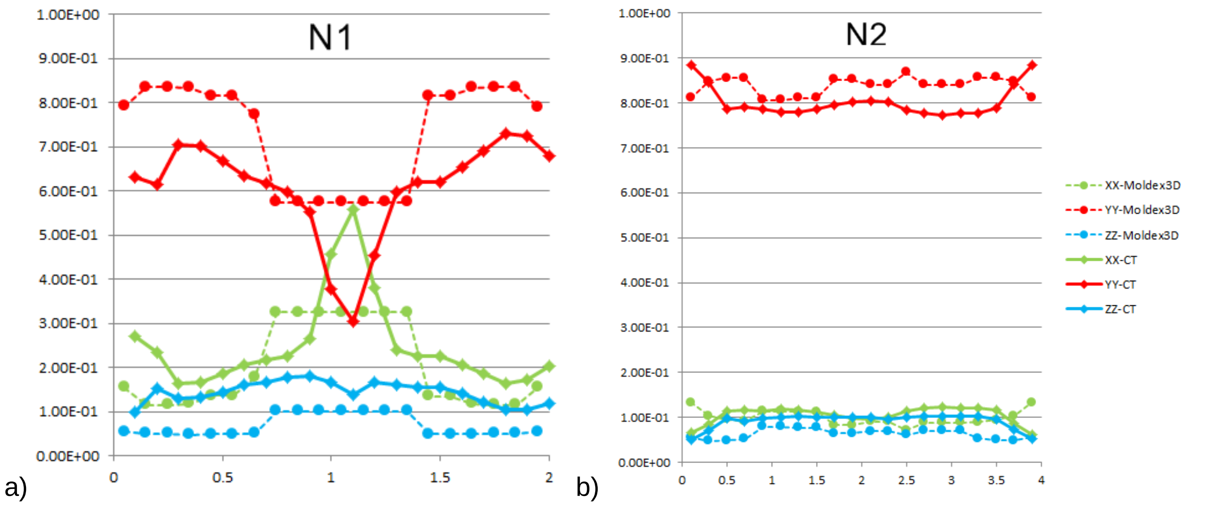

Fig.3: Degree of fibre orientation from microCT analyses and Moldex3D R14 injection molding simulations for the fibre-orientation specimens N1 and N2.

4a fibermap was used to map the second-order fibre-orientation results from the injection-moulding simulation onto a structural *LS-Dyna* mesh. For the ***MAT_215** material model ***ELEMENT_SOLID_ORTHO** and ***INITIAL_STRESS_SOLID** keywords are used to assign the local process induced fibre orientation to the elements (element system and eigenvalues). For the structural simulations of the fibre-orientation specimens, hexahedral elements of type 1 were used. The finite-element-model results were than compared with experimental quasi-static tensile-testing results (Fig.4). The anisotropic model was able to capture the difference between the two tensile specimens. However, the stiffness of both samples was overestimated by about 10%, which can be explained either by a high Young's modulus (the material model is optimised for high strain rates and overestimates the stresses at low strain rates), a higher fibre orientation value out of the injection-moulding simulation (N1) or by a change of the fibre aspect ratio due to fibre breakage during injection moulding.

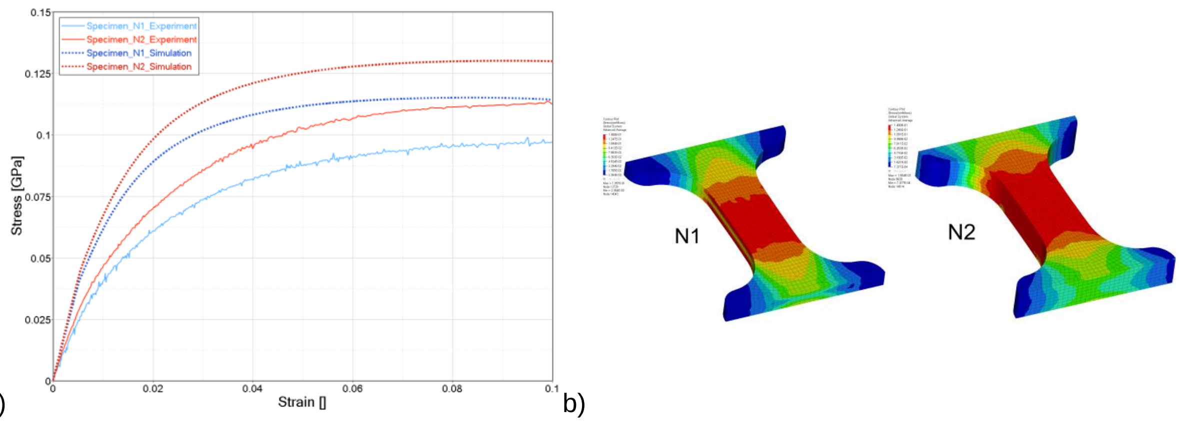

Fig.4: a) Stress–strain curves and b) simulated von-Mises stress fringe plots of fibre-orientation specimens during tensile testing. The formation of a mid-layer in specimen N1 leads to a reduced stress level in the core of the sample.

Fibre-length measurements have shown that fibre aspect ratio in the fibre-orientation specimens was significantly smaller than in the plates used for the material characterisation. The plates show an average aspect ratio of 30, the fibre-orientation specimens about 25. The mechanical simulation was done now with the reduced aspect ratio. Additionally, a static Young's modulus and the plastic strain curves were adjusted to better fit the material behaviour at low strain rates. Initial stiffness and the final stress level have significantly improved (Fig. 5). It has to be pointed out that these adjustments need deeper investigation before a final conclusion can be made. It shall be stated at this point that the fibre aspect ratio significantly influences the mechanical behaviour and needs to be considered when parameterising a micromechanics material model.

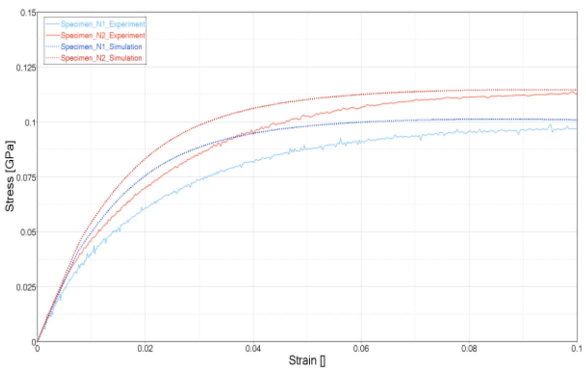

Fig.5: Stress-strain curves of the fibre-orientation specimens with simulated data with the modified material model accounting for the reduced fibre aspect ratio and the low strain rate.

4 Analysis on Component

In the third phase, the benefit of integrative simulation was investigated by comparing component level finite element simulations with results from an isotropic material model and from experimental tests. An injection-moulded component made of the same PA6 GF30 as the specimens above was tested under impact-compression loading and filmed using a high-speed camera to capture the failure mode. The test was performed on a drop-tower, with a drop-weight of 7 kg and an impact speed of 3.1 m/s. The resulting force is measured on the lower support of the specimen using a force transducer. Like before, the injection-moulding results as shown in Fig. 6 were used and transferred to the structural mesh.

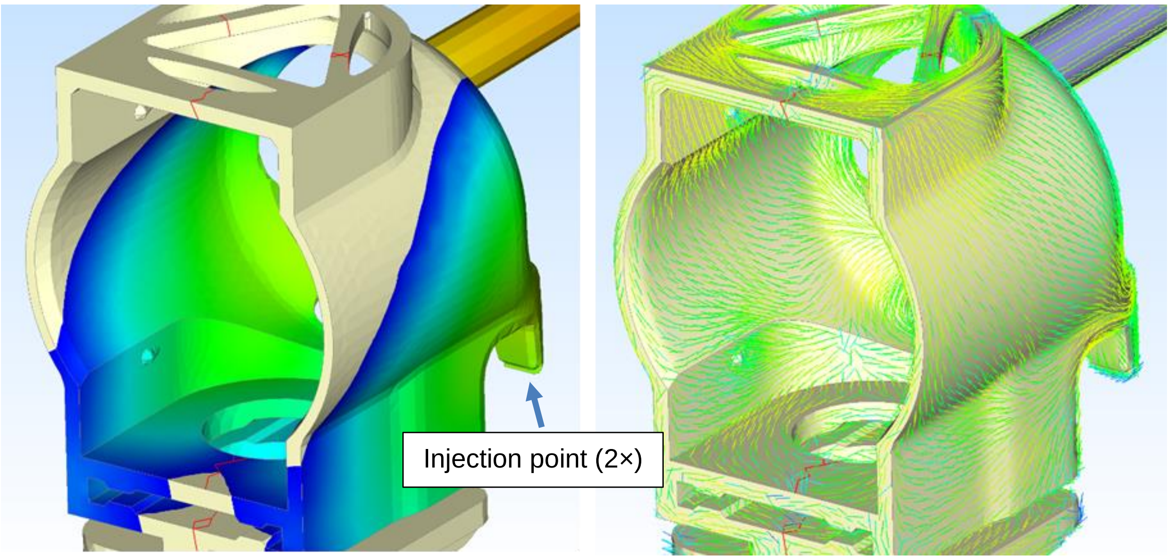

Fig.6: Pictures of the mould flow simulation results, showing the injection point, the melt front and the surface fibre orientation.

For comparison, an isotropic model was generated using ***MAT_24** based on longitudinal tension tests for different strain rates. Finite-element simulations showed that the isotropic model failed to predict the correct location of fracture onset. In contrast, the anisotropic model could accurately predict the failure mode of the component (Fig. 8).

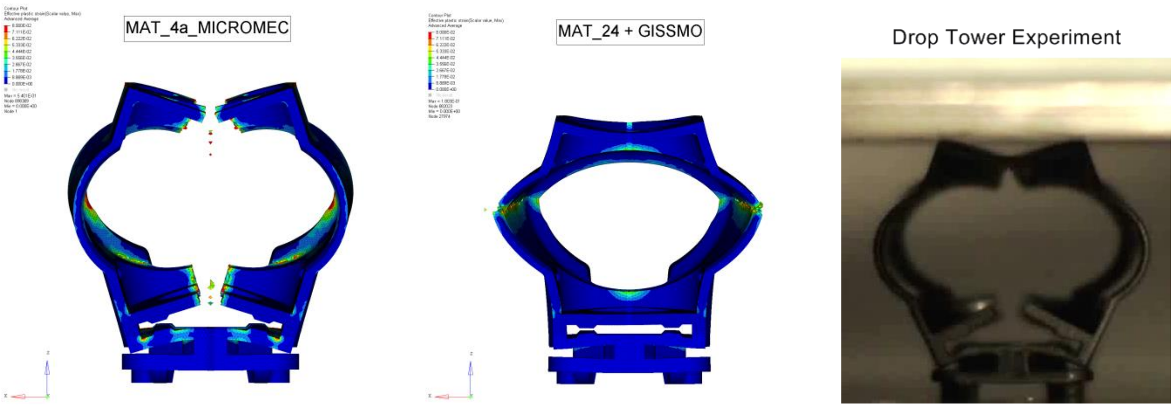

Fig.7: Failure mode of ***MAT_4a_MICROMEC** *model considering local anisotropies and isotropic* ***MAT_024** *Model in comparison with experimental drop tower experiment.*

The comparison of the forces curves of experiment and simulation show a significant overestimation by the isotropic material model. This is explained by the fact that the model is calibrated for loading of the material in flow direction, but the fibre orientations in the component at hand are not oriented in the direction of loading because of the lateral injection point. The anisotropic model accounts for this disadvantageous fibre orientations and shows a significantly better load-level prediction. Onset of failure occurs at a reasonable time. However, it has to be noted that the decrease of the loads at about 0.5 ms cannot be reproduced by the simulation and simulated failure times have to be interpreted carefully. The

reasons for the still existing deviations may be manifold: the weld line is highly loaded in the prevailing deformation mode, and its behaviour is possibly still not described correctly by the current method. Another reason may lie in friction, which may influence the force level especially at larger deformations after first failure. Further analysis of this is needed on these topics.
The model was also simulated with solid element type 4 instead of 10, which is a quadratic element formulation. This might reduce the forces even further due to the low number of only three elements over thickness in this model, which is commonly too low for bending situations.

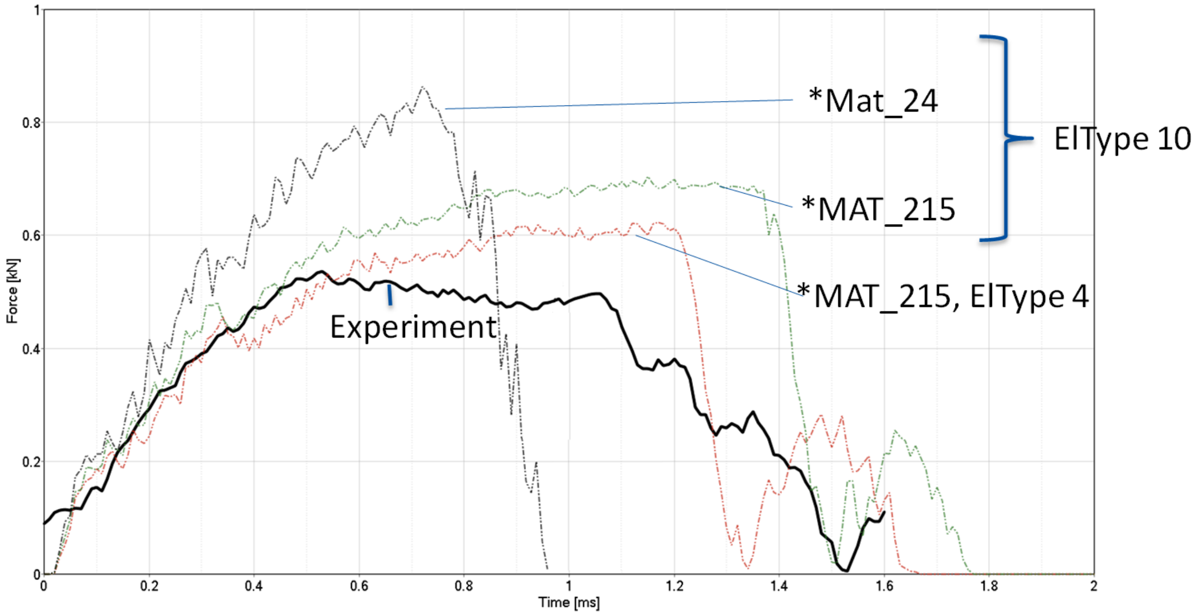

Fig.8: Force-time curves of the drop tower test on the component from experiment and simulation with different material models and element types.

5 Conclusion and Outlook

The integrative simulation approach was successfully implemented in LS-Dyna using Moldex3D for the injection-moulding process simulation and 4a fibermap for mapping the fibre orientations on the structural model. An improvement in the prediction of the failure behaviour and the load levels could be achieved compared to an isotropic `*MAT_024` model.
An important aspect is that the aspect ratio can significantly influence the material behaviour and needs to be considered.
Further work will be necessary to better understand the influence of the weld line. The development of suitable tools may be useful to map the weld line positions onto the structural model.

6 Acknowledgement

Test data used in this study were mainly coming from a research project with PCCL and HILTI. The authors would like to thank Daniel Tscharnuter, Matthias Morak, Wolfram Hahn and Michael Göttlinger for the work in that project and for building up knowledge in the field of material characterisation and modelling.

[1] P. Reithofer et al., *MAT_4a_MICROMEC – Theory and Application Notes, 2017, *Conference Proceedings 11th European LS-Dyna Conference*, Salzburg
[2] ISO 1110:1995, Plastics -- Polyamides -- Accelerated conditioning of test specimens
[3] P. H. Foss et al. 2014, Prediction of Fiber Orientation Distribution in Injection Molded Parts Using Moldex3D Simulation, *Polymer Composites*, 35 (4): 671–680