DESIGN METHODOLOGIES FOR AUTOMOTIVE COMPONENTS MADE FROM SHORT CARBON FIBER REINFORCED THERMOPLASTIC MATERIALS

Martin Reiter[1+], Andreas Pröll[1], Markus Thurmeier[2] and Zoltan Major[1]

[1]Institute of Polymer Product Engineering, Johannes Kepler University Linz, Austria,
[2] Audi AG, 85045 Ingolstadt, Germany
[+] martin.reiter@jku.at

Keywords: short carbon fiber, recycled carbon fibers, micromechanics simulation, automotive

ABSTRACT

Short carbon fiber reinforced thermoplastic materials, which are produced from recycled carbon fibers, offers new possibilities for lightweight design in high-quantity products. In this study a development process of a highly loaded automotive component made from such materials is presented. The simulation methodologies and design principles are described. Finally the developed component was produced by injection molding and mechanically tested. Validations of the simulations were performed and comparisons between conventional short glass fiber materials and these novel materials were drawn.

1 INTRODUCTION

The increasing demand and production of carbon fiber composites raises questions about their possible recycling, especially because carbon fiber waste poses a high valuable resource for novel materials. One possible application is the compounding of the processed fiber waste in form of short fibers into a thermoplastic matrix. In this paper the design and the development strategies of a structural automotive component made of these novel materials is presented. Using such materials the superior strength of carbon fibers can be combined with the processing advantages of injection molding. For a proper design various micromechanics based simulation approaches are already available and industry standard for the stiffness and strength prediction of short fiber reinforced materials.

2 TASK

The main goal of this work was the development of a novel transmission mount for passenger cars, which is made of injection molded short carbon fiber (scf) reinforced Polyamide that can be optionally reinforced with unidirectional carbon tapes. Various load cases for static loading were given, whereas the focus was put on load case 1, which seemed to be the most critical case for the design. Load case 1 was defined by a +-20kN force in Z-direction and a maximum allowed deformation of 3mm under that load. In addition to the load cases, a well-defined building space was given by the surrounding components, which limited the design freedom.

2.1. Material

For this study a scf material with 40w% carbon fibers was selected. The material grade Akromid A3 ICF40 (Akro-Plastic GmbH) which is based on recycled fibers was selected. As a reference material a typical Polyamid with 30% glass fibers Weromid PA 66 GF 30 (WMK Plastics GmbH) that is commonly used in mechanical applications was selected. The reference material was chosen in order to validate the simulation of the filling process and the mechanical simulations.

2.2. Requirements

Various load cases for static loading were given, whereas the focus was put on one specific load case, which seemed to be the most critical case for the design. A +-20kN force in Z-direction is applied and a maximum allowed deformation of 3mm must not be exceeded. This focuses the design of the component on the maximization of stiffness, while reducing the weight as much as possible. In addition to the all given load cases, a well-defined building space was given by the surrounding components, which limited the design freedom. Another primary goal was the reduction of weight to below 2.5kg, while focusing on the static requirements for stiffness and strength. Secondary requirements such as fatigue life time and acoustic properties could only be estimated due to the lack of proper material data and were therefore not in the main focus of this development.

3 METHODOLOGY

3.1. Simulation approaches

Two simulation approaches were chosen corresponding to the finalization stage of the component. In the beginning of the development, where little information about the final geometry and anisotropic material data were available, finite element simulations with isotropic linear elastic as well as elastic-plastic material models were performed. A basic design was generated using these simulations. After fiber orientation dependent material data were available and the isotropic simulations lead to a first design concept an integrated micromechanics simulation was performed using the software tool DigimatCAE (MSC Software Corp.). Therefore, an injection molding simulation with assumed injection points was conducted with Moldex3D (CoreTech System Co.). The fiber orientation tensors were extracted, mapped to the finite element mesh and a fully coupled simulation was performed using the software packages DigimatCAE and Abaqus (Dassault Systèmes). The deviation between the isotropic model and the model with local anisotropy was determined and the design was optimized accordingly.

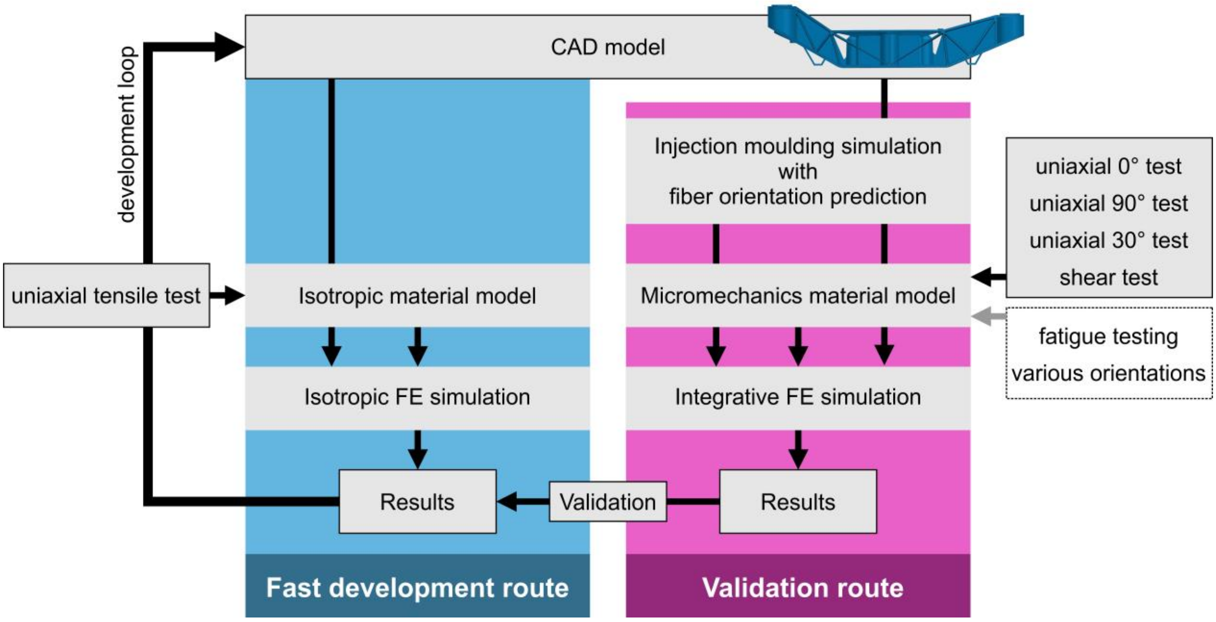

Figure 1: Simulation and product development strategy considering a fast isotropic approach and a detailed anisotropic micromechanics approach.

3.2. Design strategies

Both simulation routes the fast isotropic and the more detailed anisotropic approach were followed in parallel as depicted in Fig. 1. The main design decisions are driven by isotropic simulations that are performed for all design iterations, whereas the fully coupled anisotropic simulations are only conducted for certain selected geometries. These detailed simulations are used for validation of the global stiffness response of the isotropic approach and give additional insight into local design features. Some local features, for example insert geometries, were packed into sub tasks, which were handled independently from the global component design. In this paper the focus is put on the overall component design and the different simulation methodologies.

In order to acquire the most innovative solutions within this task creative design strategies such as morphological boxes and mind maps were applied. Although these design approach are not focus of this paper, one design decision is worth mentioning. It was expected that a macroscopic sandwich-like (double-decker) structure reveal both higher stiffness, better light-weight ratio (LWR) and higher strength that a conventional one-side open rib reinforced structure. Hence, in-spite of the significantly larger complexity of the component geometry and the complexity and demand on the molding tool (incl. costs), this solution was preferred and followed through the further development process. The large free surfaces of both outer walls make the application of UD reinforcing tapes also possible. The very good LWR ratio can even be significantly improved by a proper positioning and consolidation of the tapes within the injection molded structure.

4 RESULTS

In this study a fast isotropic simulation approach and a more detailed anisotropic simulation approach based on micromechanics were chosen and conducted in parallel. In the following the amount of time to finish one complete iteration loop is discussed. Starting from an altered CAD geometry the two simulation routes are followed up to the evaluation of the results.

4.1. Material data and model calibration

Before the first iteration loop can be started, adequate material data for the various material models have to be determined.

In order to set up the isotropic material model based on elastic plastic assumptions uniaxial tensile tests using ISO 527-2 multi purpose specimens were conducted at different testing rates. One must keep in mind that the degree of fiber alignment might be higher in a standard tensile specimen than in the actual component, which will lead to an overestimation of the true physical properties. The first component a_{11} of the fiber orientation tensor of an ISO 527 multi purpose specimen was determined to be 0.86 according to Hartl et al. [1] in a PP-SGF material. Whereas in comparison this value can reach a minimum of 0.53 in injection molded plates [2], which indicated also a plausible orientation in sheet like components.

For a proper calibration of a micromechanics based material model a significant higher experimental effort is necessary. In order to account for the anisotropy tensile test of at least three different orientations are necessary. In this study injection molded tensile specimens with well controlled fiber orientations were produced and tested under uniaxial loading. These specimens were produced using a special injection molding tool (see Figure 2), which allows for high fiber orientations due to its small thickness of 2mm and the specially designed melt flow acceleration zones. Specimens with preferred orientations of 0°, 30° and 90° were milled from the whole injection molded specimen component. Consequently, uniaxial tensile tests were performed at various testing rates and a micromechanics based material model was calibrated using DigimatMX. In this case, the matrix material was assumed to follow an elastic-viscoplastic behavior. This model calibration could be principally also performed on tensile tests of specimens that were cut from injection molded plates [3], [4]. However, it must be emphasized that plate specimens reveal a poor degree of fiber orientation compared to the proposed specimens, which might lead to a less reliable calibrated micromechanics model.

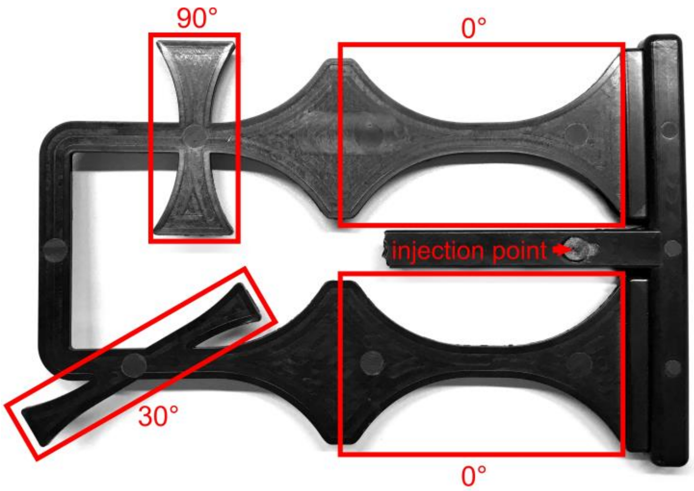

Figure 2: Picture of the injection molded unidirectional 0°, 30° and 90° specimens.

4.2. Isotropic simulation approach

For the isotropic simulation the FE solver NX Nastran (Siemens PLM) was used. The CAD geometry was designed and adapted in Siemens NX (Siemens PLM). The first step was the meshing of the component, which took in this specific case usually between 4h and one day depending on the time spend on defeaturing and simplifying the CAD component as well as on the level of local mesh refinement. After the revised geometry was remeshed, also the boundary conditions, loadings and interactions had to be reassigned to the correct geometry features.

In this simulation route the software environment of Siemens PLM was chosen, because associativity could be sustained between the CAD part and the simulation model, which drastically decreased the amount of time that was necessary to rebuild the simulation model. In an optimum case, where the changes to the CAD part were marginal and associativity could be mostly kept, the simulation model could be restarted within an hour.

The NX Nastran solver ran one load case of the isotropic model with 1.4 mio elements in approximately 20min on 4 cores. With this performance – especially if full associativity between CAD data and simulation model was available – one to two iteration loops could be run per day. A detailed listing of the process times of this workflow is shown in Figure 3.

4.3. Anisotropic simulation approach

In our anisotropic simulation approach no associativity between CAD and simulation model was available, which means that the defeaturing and meshing of the component had to be started for each iteration from scratch.

At first a possible manufacturing process including a coarse mold design, proper injection points as well as processing parameters had to be assumed. The re-setup of the injection molding simulation model after a change in geometry took on average four hours. With a computation time of approximately five hours, this process could be finished within one day.

The next step was the re-building of the FE model. The most time was consumed by the meshing of the component. Since associativity between the applied FE pre-processor and the CAD software was not available, the mesh including all local mesh refinements and mesh enhancements had to be redone, which resulted in a re-modeling time between a have and a full day. At this point it must be emphasized that all times are taken on basis of this specific component. These times can drastically vary if the component reveals more or less local features, a different complexity or features that are generally complicated to mesh.

Before the simulation can be started an additional step of fiber orientation mapping is necessary in order to assign each element of the FE mesh a proper orientation tensor. In our case, this step was performed within Moldex3D. After mapping was performed the integrative simulation was started.

The software tool DigimatCAE allows to run the finite element simulation in fully micromechanically coupled way, where the homogenization problem is solved for each element in every iteration. This type of coupling drastically increases the computational effort. As it can be seen in Fig. 3, the computational time increased by a factor of ~50x in this setup. In addition to this fully micromechanical coupling a hybrid method is provided by DigimatCAE [5]. Using this approach the anisotropic behavior is precomputed over a wide range of orientations and a mathematical fit model is used during the FE solving. By increasing the computational costs only by a factor of 1.7 instead of 50, the hybrid approach yields to a significant speed up.

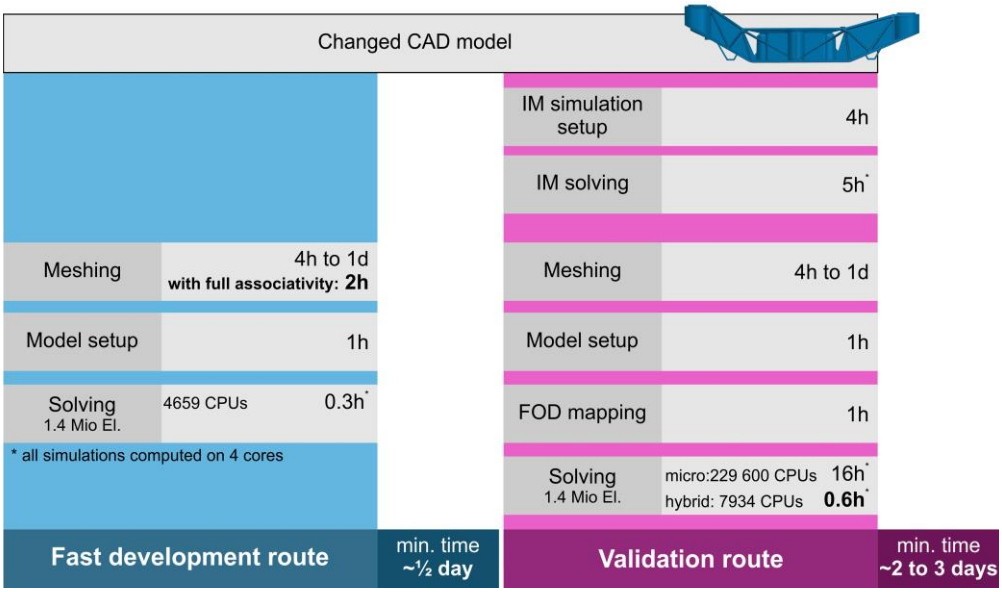

Figure 3: Time and computational effort of one development iteration cycle starting from a revised geometry model. Comparison of the isotropic simulation approach and a fully coupled anisotropic simulation.

Using the coupled hybrid approach one iteration cycle can be performed in approximately two to three days. Comparing the two simulation routes it can be seen that the isotropic approach shows an advantage in time by a factor of minimum four to six when re-computing a revised geometry model. One must also keep in mind that the coupled simulation approach requires a significantly higher experimental effort in order to achieve reliable material models. But once these micromechanic models are available, this methodology allows for a higher result accuracy as well as a more detailed insight into the deformation behavior. A comparison of the von Mises stresses computed by the isotropic and anisotropic model are shown in Fig. 4. One can clearly see that the asymmetric stress distribution resulting from the position of the injection points cannot be reproduced by the pure isotropic model. If not considered correctly, local features such as weld lines can lead to unexpected failure, if they are located in highly loaded regions.

All primary design decisions were made based on the isotropic model that was computed for each design iteration. Using this model, rib structures and general design considerations could be quickly evaluated. In order to achieve reliable results out of this model, the model stiffness was calibrated based on the anisotropic coupled simulations. In addition, mechanically significant details were also designed based on the anisotropic model.

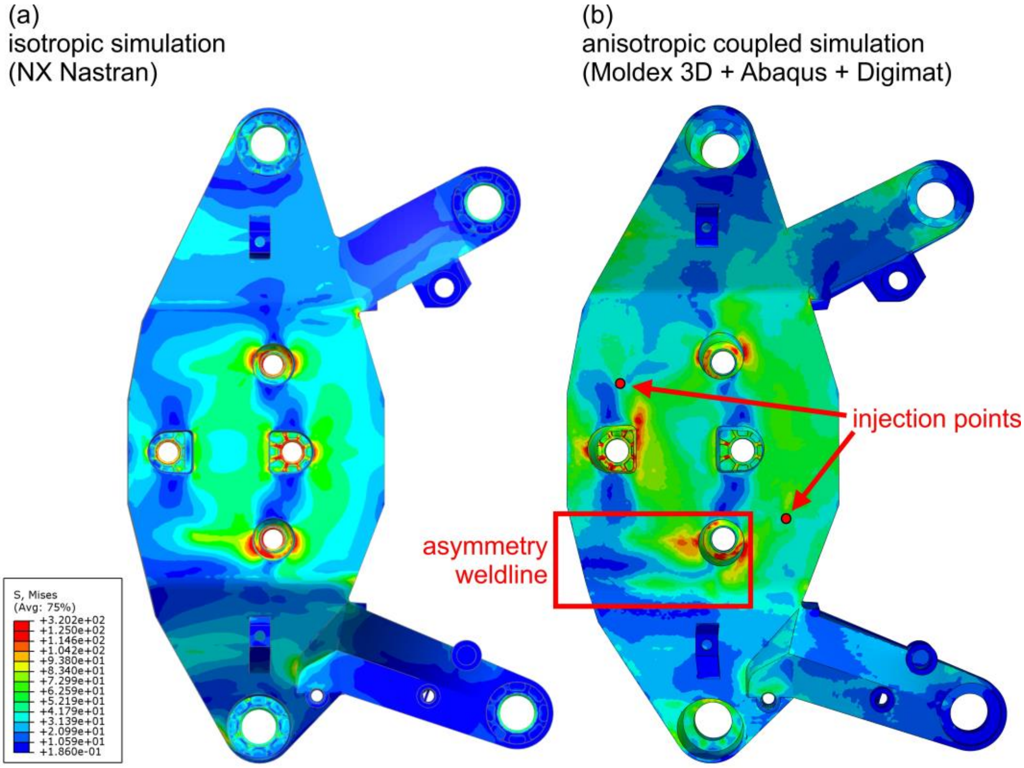

Figure 4: Comparison of the computed von Mises stresses under the specified load case. (a) isotropic simulation model using NX Nastran solver; (b) Anisotropic fully coupled simulation using Moldex3D, Abaqus standard and Digimat.

4.4. Final design and preliminary tests

After several iterations the use of both simulation strategies the isotropic and fully coupled simulation lead to a final component design (Fig. 5a). A suitable injection molding tool was manufactured (Schöfer GmbH, Austria) and the component was produced with different materials. In a first shot a commonly used material grade PA66 with 30%vol short glass fibers (sgf) and the novel material compound PA66 with 30%vol scf were injected.

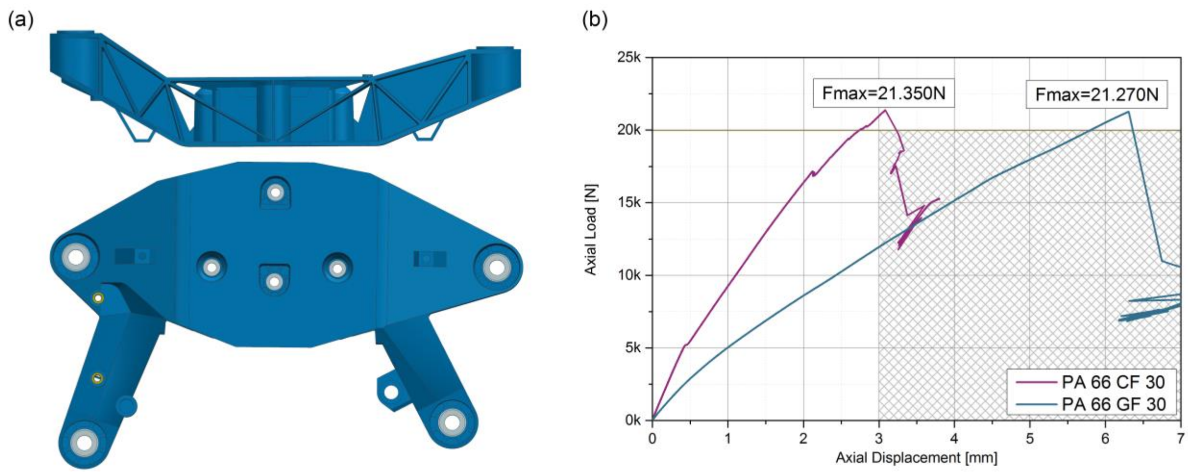

Figure 5: (a) CAD geometry of the injection molded component. (b) Force displacement curves of component tests at load case 1. Comparison of PA66-CF30 and PA66-GF30 material.

Thus, the glass fiber and carbon fiber components were tested under monotonic loading until failure (Fig. 5b). It can be seen that the scf-components revealed a significant higher stiffness than the sgf-components. Moreover, using the scf-material the required stiffness of 20kN within a 3mm displacement could also be achieved. The strength on the other hand showed a similar value for both material grades.

Using the scf-material an overall weight of 1.62kg could be achieved, which is significantly below our target of 2.50kg. In comparison to the sgf-component which reveals a mass of 1.88kg, a 14% reduction in weight could be realized.

9 CONCLUSIONS

It was shown in this study that scf reinforced materials using recycled carbon fibers can lead to a significant improvement in component stiffness and moreover to tremendous increase in the stiffness to weight ratio. However, the drawback of this novel material class is that component design has to be performed with care because of the relatively low failure strains and the high sensitivity to local stress intensities. But, if all relevant factors are considered, these materials offers the possibility to furtherly improve lightweight design in high-quantity parts.

To speed up the development process, each design iteration was simulated by a purely isotropic approach that could be conducted at maximum speed within a half day. In order to keep this simple simulation reliable and validate all design features, a coupled simulation approach was used, which combined processing, microstructure and structural simulation. After each major geometry revision, a coupled simulation a performed and the stiffness response of the isotropic simulation was calibrated. All crucial features were designed based on the anisotropic coupled simulation.

At this time only preliminary component tests have been available. In an ongoing work different injection molding materials will be compared. Fatigue testing as well as fatigue simulations will be performed in order to elaborate new methodologies for the fatigue prediction of short fiber reinforced materials. Furthermore, the design of the component was done in a way that additional UD reinforcement tapes can be placed at top and bottom to further enhance the mechanical behavior.

REFERENCES

[1] A. M. Hartl, M. Jerabek, D. Salaberger, M. Reiter, Z. Major, and R. W. Lang, "Experimental approach to characterize the orientation anisotropy in the tensile stress-strain behavior of short glass fiber reinforced polypropylene," *Compos. Part A Appl. Sci. Manuf.*, 2014.

[2] A. M. Hartl, M. Jerabek, P. Freudenthaler, and R. W. Lang, "Orientation dependent compression/tension asymmetry of short glass fiber reinforced polypropylene: Part 1- Macroscopic deformation and failure," *Compos. Sci. Technol. or Compos. Part A*, no. Part 1, pp. 1–24, 2014.

[3] A. Bernasconi, P. Davoli, A. Basile, and A. Filippi, "Effect of fibre orientation on the fatigue behaviour of a short glass fibre reinforced polyamide-6," *Int. J. Fatigue*, vol. 29, no. 2, pp. 199–208, Feb. 2007.

[4] K. Tanaka, T. Kitano, and N. Egami, "Effect of fiber orientation on fatigue crack propagation in short-fiber reinforced plastics," *Eng. Fract. Mech.*, vol. 123, pp. 44–58, Jun. 2014.

[5] "Digimat 5.0.1 Manual." e-Xstream engineering SA, Louvain-la-Neuve, Belgium, 2013.