# On the development of a finite element model to analyze the behavior of hybrid composites considering the manufacturing history

**M Riemer[1], R Kießling[2], R Müller[1], W-G Drossel[1], D Landgrebe[1], J Ihlemann[2]**

[1] Fraunhofer Institute for Machine Tools and Forming Technology IWU, Reichenhainer Straße 88, 09126 Chemnitz, Germany
[2] Chair of Solid Mechanics, Faculty of Mechanical Engineering, Chemnitz University of Technology, Reichenhainer Straße 70, 09126 Chemnitz, Germany

E-mail: matthias.riemer@iwu.fraunhofer.de, robert.kießling@mb.tu-chemnitz.de

**Abstract**. Hybrid parts are strongly moving into the focus for lightweight applications. Unfortunately, the accurate, simulative design, which comprises the accurate prediction of final part geometry, is still a challenging task. In the scope of this paper, an approach to improve the accuracy of appropriate finite element simulations is presented. To this end, the manufacturing history of the hybrid part is considered within the simulation of the part behavior. To create a finite element model of the considered hybrid composite, the intrinsic manufacturing process is modelled first. This includes the modelling of the thermoforming process of a fiber reinforced polymer as well as the sheet metal forming process for the fabrication of form fit elements. Then, the geometry of the hybrid part is deduced from the geometries of the single components. Afterwards, the material properties, including the local fiber volume content as well as the local fiber orientation, are mapped to the finite elements. Consequently, a workflow to create a finite element model which considers manufacturing history is developed and successfully tested.

## 1. Introduction

Hybrid parts, which combine the advantages of different material classes, are more and more utilized for lightweight applications [1]. These materials are made of different material classes to combine the advantages of the applied materials [2]. In the field of crash relevant structures, the combination of fiber reinforced plastics (FRP) with highly ductile metals seems to be advantageous. FRPs with their excellent strength-to-weight ratio carry operating loads. The ductile metal prevents brittle material failure behavior under crash loading. The design of hybrid parts offers a lot of freedom regarding, for example, the layer structure, the metallic material or the kind of reinforcement. Many variants of the part are feasible. Consequently, an accurate numerical prediction of the mechanical behavior is essential for an efficient product development process. However, the simulative design of hybrid parts is still a challenging task. One way to improve the accuracy of simulations is to account for the effects of the manufacturing process, since the manufacturing operations highly affect the performance of the produced part. This is a common approach in the field of metal forming [3, 4]. In this field, the effective plastic strain, the thickness distribution and the local damage variable are mapped from forming simulations to the crash model. Thus, these influences are considered in the crash analysis. With respect to hybrid parts under high dynamic loads, some simulative investigations are presented by [5, 6]. Nevertheless, the forming history is not considered.

This contribution describes how simulations of the manufacturing and mechanical simulations of the resulting part can be connected for the case of hybrid parts. To create an appropriate finite element model of the considered hybrid composite, the hybrid manufacturing process is modelled first. Herein, the forming simulation of the FRP component and the forming simulation of the metallic part are separately realized. Then, the geometry of the hybrid part is estimated from the geometries of the single components. Afterwards the material properties, including the local fiber volume ratio or the local fiber orientation among others, are mapped to the integration points of the finite elements. Thus, a highly complex finite element model of a hybrid composite is developed. This model enables the evaluation of the influence of the manufacturing history on the mechanical behavior of the considered composite.

## 2. Intrinsic manufacturing of a hybrid composite

The investigated hybrid part consists of a continuous FRP, in which a metallic reinforcement structure is integrated. The FRP is a unidirectional carbon fiber sheet (thickness of 1.3 mm) composed of a thermoplastic PA 6 (BASF Ultramid B40) and Toray T700S (50-K) carbon fibers. The metallic layer is made up of AA 5182 and has a thickness of 0.2 mm. The hybrid part is manufactured in a so called 'intrinsic process'. The intrinsic approach combines the shaping of the part as well as the joining of the metallic insert and the FRP in a single process step [7]. The performance of a hybrid part strongly depends on the interface between the different materials. To improve interface strength, a combination of adhesive bonding and mechanical form fit is utilized. In contrast to state of the art processes, the form fit elements are generated within the intrinsic manufacturing process. To this end, a special metallic insert is developed and the manufacturing is simultaneously performed on two geometric scales (see, for details, [8]). On the macro scale, a conventional thermoforming process for shaping the part geometry is applied. On the mesoscopic level, the mechanical form fit elements are formed by applying a tension load on the metallic insert (cf. Figure 1). The local forming process can be classified as a sheet metal forming process. During the further global forming, the form fit elements are pressed in the molten matrix of the FRP and realize the mechanical interlocking. For the combination of the different sub-processes in a single step process, a variothermal forming tool is used [9]. The raw material is directly heated up in a pre-heated forming tool until the forming temperature of the thermoplastic matrix is reached. During the global and local forming process the tool temperature is constant. The consolidation phase is started by cooling the tool when the global forming process is finished.

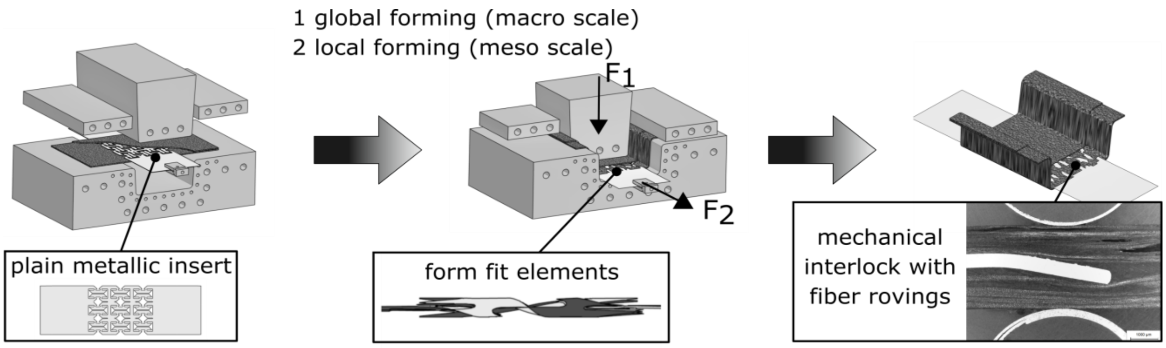


**Figure 1.** Substeps of the investigated intrinsic process chain [9].

Keuthage et al. [10] show that the mechanical properties of a hybrid part depend on the fiber volume content, the fiber orientation, the local thickness of the FRP and the geometry as well as the effective plastic strain of the metallic insert. Within this contribution, manufacturing history variables are identified within separate simulations of the forming processes. In the following section, these simulations of the local and the global forming process are presented.

IOP Conf. Series: Materials Science and Engineering **418** (2018) 012128 doi:10.1088/1757-899X/418/1/012128

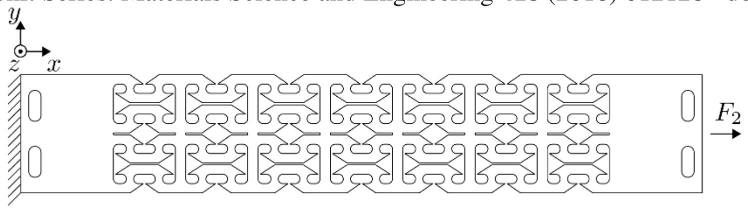


**Figure 2.** Geometry of the metallic insert and the applied boundary conditions [11].

## 3. FE modelling of the intrinsic manufacturing process

### *3.1. FE simulation of the deformation of the metallic insert*

Within the manufacturing process, the metallic insert is plastically deformed to generate the form fit elements. To estimate the resulting geometry, a FE simulation of this forming process is carried out. Due to the characteristic of the forming, a post buckling analysis has to be performed. Generally, a post buckling analysis consists of three steps. At the beginning, the first eigenmode is identified with respect to the boundary conditions (see Figure 2). Afterwards, the resulting displacements, given in Figure 3, are scaled by a factor δ and mapped to the geometry to induce imperfections.

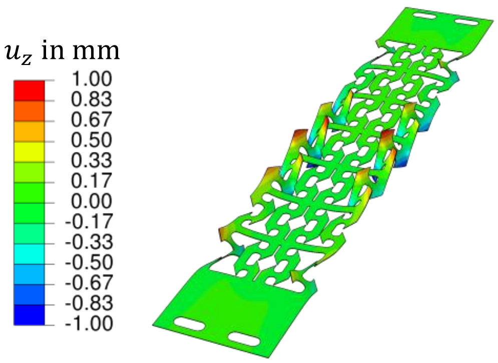


**Figure 3.** First eigenmode of the metallic insert (deformation scale factor: 10).

Finally, the forming of the metallic insert can be simulated. The according mesh consists of about 340000 fully integrated, quadratic elements. Note, that this high number of elements is required to prevent imperfections resulting from the discretization of the geometry. The viscoplastic material behavior of the applied aluminum AA 5182 is simulated by a nonlinear material model at large strains. This material model is formulated based on directly connected rheological elements (see, for details, [12]). With the help of the user subroutine UMAT, this material model is implemented in Abaqus/Standard. The required material parameters are identified from uniaxial tension tests with different time series [11]. Several FE simulations with varied factors δ are performed. The results are compared to the result of an experimental investigation of this forming process. According to the force-displacement curves given in Figure 4, a good accordance between experimental data and simulation data, obtained for $\delta = 2.0$, can be observed.

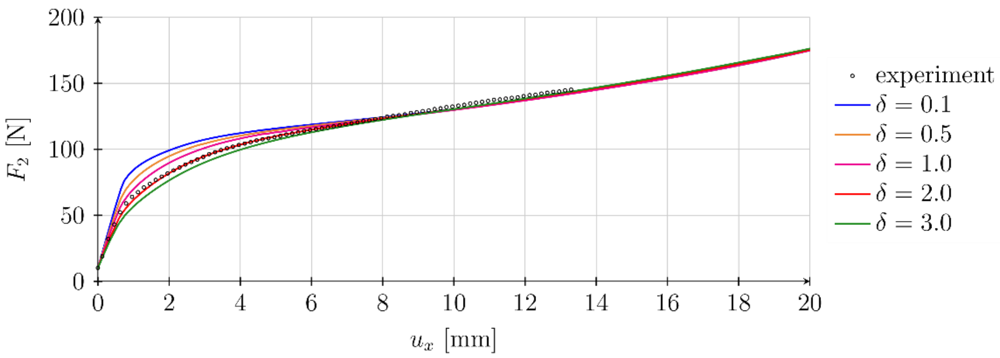


**Figure 4.** Force-displacement curve resulting from experiment and from simulations with different δ.

The force-displacement curve strongly depends on the intensity of the imperfections. Consequently, this dependence will have to be in mind if the forming process of the metallic insert is configured. However, the finite element simulations aim for the identification of the geometry of the metallic insert, especially the out-of-plane deformation of the form fit elements. In Figure 5, the out-of-plane displacement fields resulting from the FE simulation with $\delta = 2.0$ and resulting from analyzing the forming process using the optical measuring system GOM ARAMIS are presented.

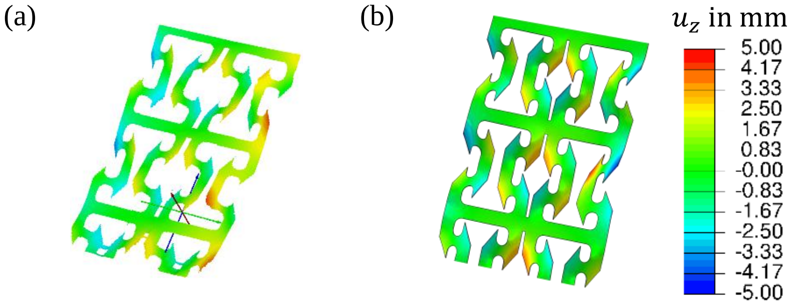


**Figure 5.** Displacement fields resulting from experiment (a) and from simulation with $\delta = 2.0$ (b) for $u_x = 9.5$ mm.

The resulting displacement fields are in good accordance keeping in mind that the initial imperfections of the FE model and the investigated metallic insert are not fully coincident. Thus, the developed FE model is suitable to estimate the geometry of the metallic insert. Hence, the influence of the imperfections on the resulting geometry can be simulatively analyzed. Figure 6 shows the comparison of the averaged out-of-plane displacements of the peaks of the form fit elements. It can be stated that the final geometry of the metallic insert is not strongly influenced by the intensity of the imperfections having the difference of the initial position of the peaks in mind. Due to this finding, the geometry resulting from a finite element simulation with $\delta = 2.0$ will be applied in the further course.

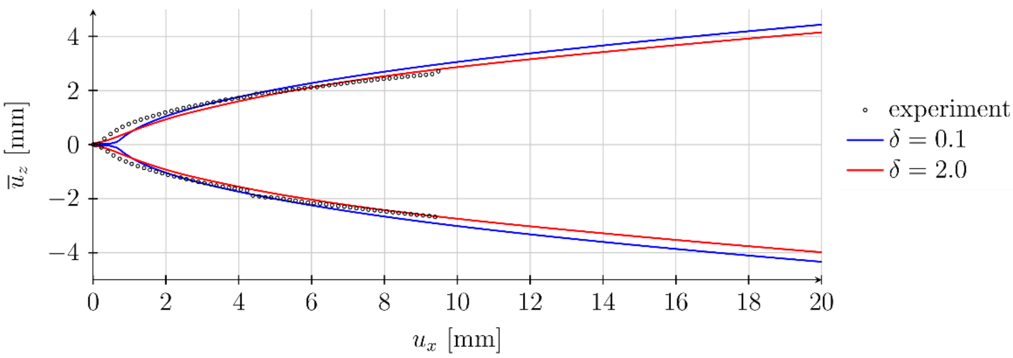


**Figure 6.** Averaged out-of-plane displacements of the peaks of the form fit elements resulting from experiment and simulations.

### *3.2. Simulation of the thermoforming*

The aim of simulating the global forming process is the prediction of the geometry, the thickness distribution as well as the local fiber volume content and direction of the FRP component. In general, there are different approaches for simulating thermoforming processes [13]. Here, the explicit FE code LS-Dyna is used to model the global forming process. LS-Dyna provides the material model *MAT_249UD for the forming simulation of thermoplastic unidirectional FRPs [14]. According to this model, the material behavior is described by an anisotropic hyperplastic material law [15]. In the scope of this work the material parameters for a unidirectional carbon fiber sheet with PA6 matrix material are taken from the literature [16, 17, 18]. Because the metallic insert is neglected in this simulation, a simple layout of two UD-layers in 0°-direction is only simulated. Figure 7 shows the geometry of the part in detail as well as the resulting model. The tool parts are represented as rigid bodies.

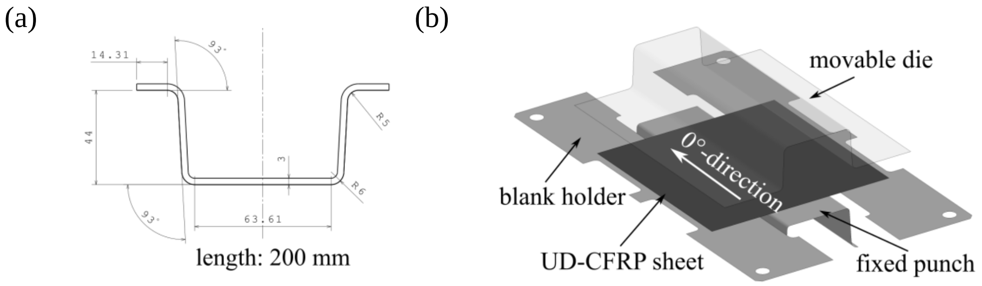


**Figure 7.** Geometry of the thermoformed part (a) and the simulation model of the thermoforming process (b).

The blank is square shaped and is 200 mm x 200 mm in size. Under-integrated shell elements with a length of 1 mm are used for the blank. The edges of the blank are free and clamped by the blank holder. The process is modelled isothermal and a forming temperature of 225 °C is assumed.
To investigate the impact of the process parameters on the part performance, the blank holder force is varied. Figure 8 shows different part properties of the thermoformed part. It is shown, that the blank holder force has a considerable impact on the thinning as well as the fiber volume content. The fiber orientation is not significantly affected.

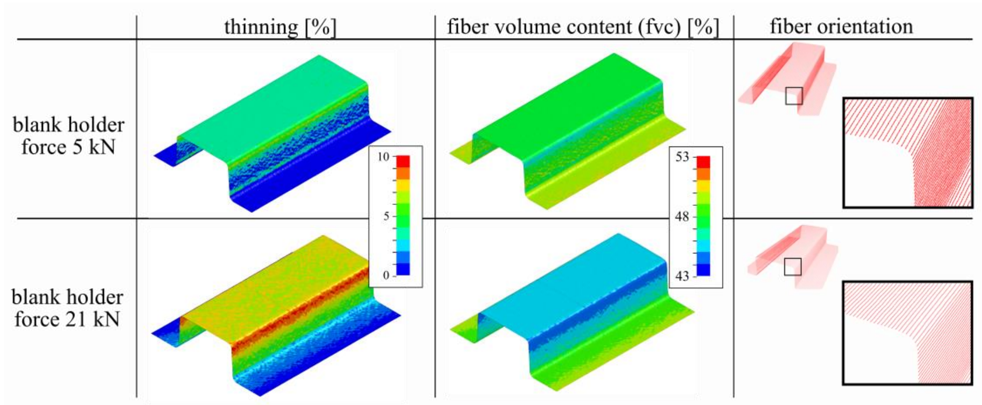


**Figure 8.** Results of thermoforming simulations with different blank holder forces.

## 4. Development of a suitable simulation model of the hybrid composite

To analyze the mechanical behavior of the hybrid composite with respect to the manufacturing history, a FE model is required and has to be developed. The mechanical behavior of the applied components is represented by appropriate, nonlinear material models at large strains. In addition to the material model utilized for the simulation of the deformation of the metallic insert (see section 3.1), an elastic-plastic damage material model and a transversely isotropic viscoplastic material model are employed for the interface and the FRP, respectively (see, for details, [8], [20]). The FE model has to include the simulation results of the thermoforming as well as of the generation of the form fit elements. Due to this, the creation of an appropriate FE model is a challenging task. Aiming for a well-shaped mesh, this is realized by using the preprocessor Altair HyperMesh and the finite element code Abaqus. At first, the geometry of the surrounding FRP is generated. Therefore, the deformed mesh resulting from the thermoforming simulation is imported in Abaqus. Note that this deformed mesh consists of shell elements. To engender a three-dimensional model, the shell elements are extruded to the calculated thicknesses and exported as a Nastran file. Afterwards, this orphan mesh as well as the orphan mesh of the deformed insert are separately imported in HyperMesh. Hence, outer surfaces can be deduced from these meshes. These outer surfaces are the basis for the further development of the FE model of the hybrid composite. The geometries of the interfaces are constructed by extruding the top and bottom outer surface of the deformed insert. The same applies for the cover plate. Here, sections of the surrounding FRP have to be extruded. Consequently, the geometries of the components are defined by surfaces. To deduce a regular mesh, the top surface of the surrounding FRP is meshed. Then, fractions of this mesh are copied and projected to other surfaces. The resulting three dimensional elements are generated by extruding the two dimensional elements discretizing the outer surfaces. Due to this meshing strategy, the different mesh components are combined by shared nodes. Consequently, no additional tie constraints has to be considered, which results in a lower computational effort. Next, the three dimensional mesh is imported in Abaqus by an input file. Within the following assembly, the meshed demonstrator is integrated for example in the simulation of a three point bending test. This comprises the definition of contact properties, among other things. Finally, the fiber data, resulting from the simulation of the thermoforming, are mapped as initial conditions of the state variables to the integration points of the finite elements of the surrounding FRP. To this end, the user subroutine SDVINI is applied. Consequently, a workflow to generate a nonlinear FE model of the hybrid composite is developed. The according steps are summarized and illustrated in Figure 9.

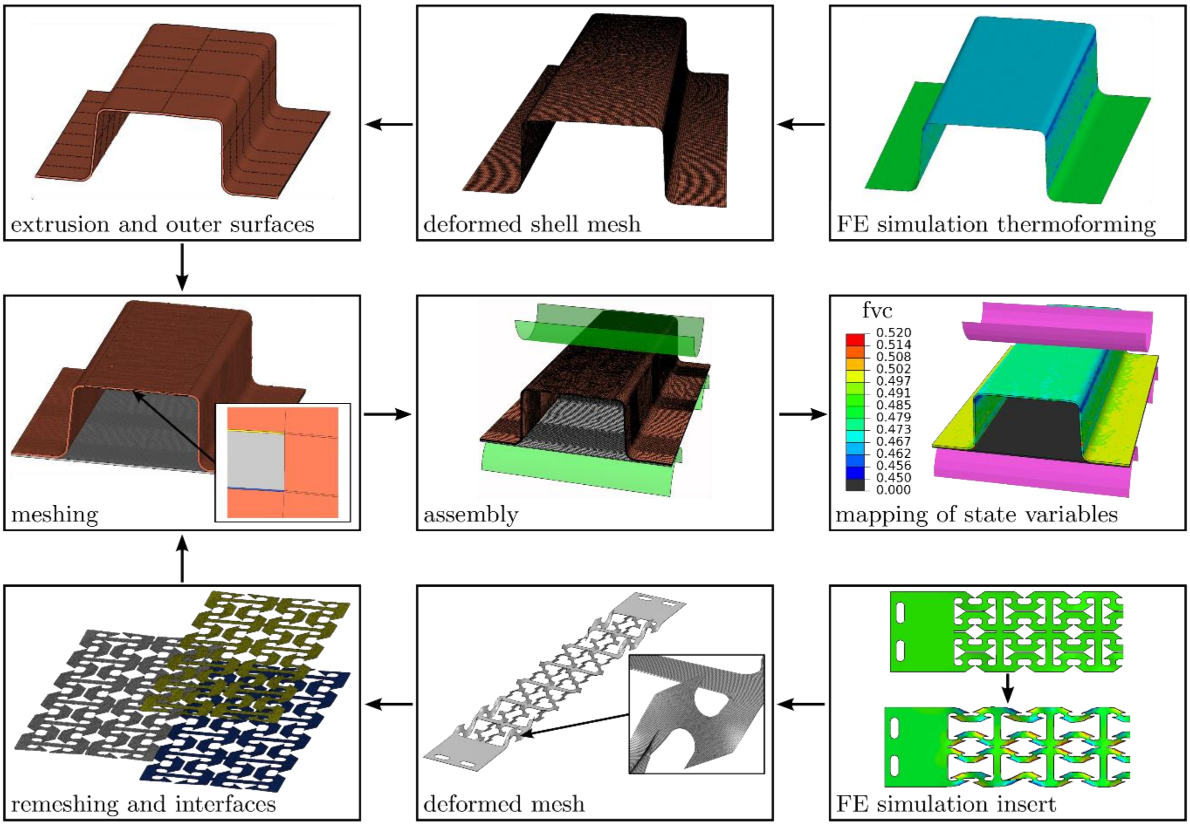


**Figure 9.** Generation of the FE model of the hybrid composite considering the manufacturing history.

Due to the utilization of the simulation results of the thermoforming as well as of the deformation of the metallic insert, the resulting FE model is able to estimate the mechanical behavior of the hybrid composite with respect to the manufacturing history. Furthermore, the effect of the mechanical interlocking is accurately modeled due the consideration of the geometry of the deformed form fit elements. Exemplarily, the results of a FE simulation of a three point bending test of the demonstrator fabricated with a blank holder force of 5kN is given in Figure 10. According to presented von Mises stress distribution, it can be observed, that the force of the loading pin is locally induced. However, the load is spread over a wider range, due to metallic insert. This can be stated from Figure 11, where the von Mises stress distribution of the metallic insert is only shown. With respect to the application of the hybrid composite for crash-relevant structural parts, this load spreading is very beneficial.

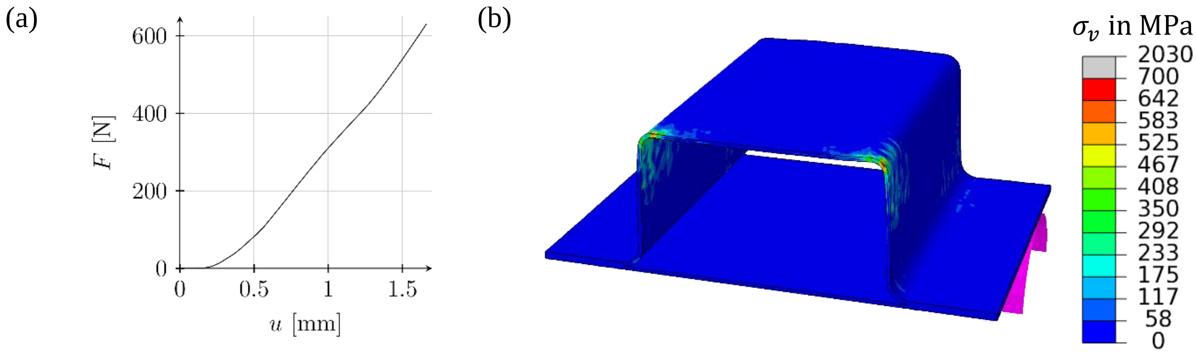


**Figure 10.** Result of FE simulation of a three point bending test of the demonstrator fabricated with a blank holder force of 5kN: force-displacement curve (a) and von Mises stress distribution for a displacement of $u = 1.5$ mm (b), cutting plane at the middle of the demonstrator.

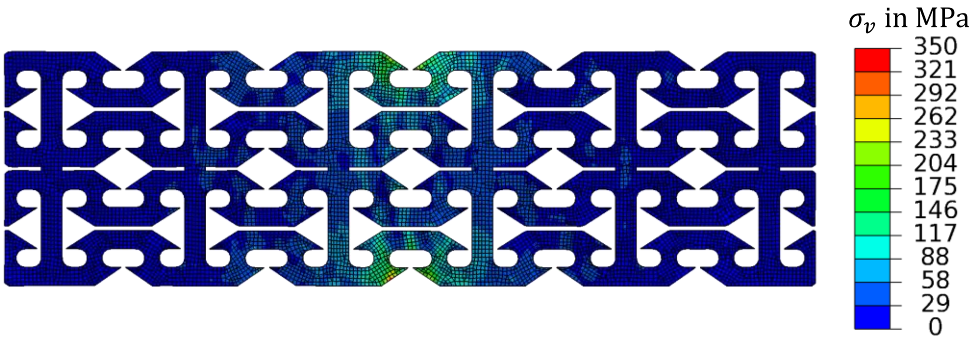


**Figure 11.** Von Mises stress distribution at the metallic insert of the demonstrator for a displacement of $u =$ 1.5 mm.

In the future, finite element models with varied process parameters will be generated and finite element simulations will be performed. Based on the results, the influence of the manufacturing history on the mechanical behavior can be virtually estimated.

## 5. Conclusion

Within this contribution, a concept for considering the manufacturing history in a FE model of a complex hybrid part is presented. After explaining the manufacturing concept of the investigated hybrid part, the modelling of the manufacturing process is presented. The intrinsic manufacturing process is divided into the forming process of the metallic insert and the thermoforming process of the FRP. Each of these sub-processes is individually modelled. The output of the manufacturing simulations are the geometry of the metallic insert as well as the geometry, the local fiber volume content and the local fiber orientation of the thermoformed FRP component. Subsequently, these data has to be transferred to the simulation model of the hybrid part. Therefore, a workflow for generating a complex FE model of the hybrid part considering the geometry and the state variables from the manufacturing simulations is developed and validated. However, the regular meshing of the complex geometry is quiet a challenging and time consuming process. In the future, some steps will be automatized by the authors. Moreover, appropriate simulations will be carried out to investigate the influence of process parameters on the mechanical behavior of the hybrid part.

**Acknowledgements**
This research was supported by the German National Science Foundation (DFG) within the Priority Program 1712.

## References

[1] Henning F and Moeller E 2011 *Handbuch Leichtbau - Methoden, Werkstoffe, Fertigung* (München: Carl Hanser Verlag)
[2] Nestler D 2014 *Beitrag zum Thema Verbundwerkstoffe und Werkstoffverbunde: Status quo und Forschungsansätze* (Chemnitz: Universitätsverlag der TU Chemnitz)
[3] Huh H, Kim K P, Kim S H, Song J H, Kim H S and Hong S K 2003 Crashworthiness assessment of front side members in an auto-body considering the fabrication histories *Int. J. Mech. Sci.* **45** 1645-60
[4] Göckler M I, Dogan U C and Darendeliler H 2016 Effects of forming history on crash simulation of a vehicle *J. Phys.: Conf. Ser.* **734** 032094
[5] Hopmann C, Klein J, Schönfuß B I, Reisgen U, Schönberger J and Schiebahn A 2017 Analysis and specification of the crash behavior of plasitcs/metal-hybrid composites *Prod. Eng. Res. Devel.* **11** 183-93
[6] Hopmann C, Schöngart M, Weber M and Klein J 2015 Crash simulation of hybrid structures considering the stress and strain rate dependent material behavior *AIP Conference Proceedings* **1664** 050007
[7] Koch S-F, Barfuss D, Bobbert M, Groß L, Grützner R, Riemer M, Stefaniak D and Wang Z. 2016 Intrinsic Hybrid Composites for Lightweight Structures: New Process Chain Approaches *Adv. Mat. Res.* **1140** 239-46
[8] Kießling R, Ihlemann J, Riemer M and Drossel W G 2016 Production and modelling of an intrinsic hybrid metal composite for automotive parts *Int. J. Auto. C.* **2** 209–28
[9] Riemer M, Müller R, Drossel W G and Landgrebe D 2017 Process development and tooling design for intrinsic hybrid composites *J. Phys.: Conf. Ser.* **896** 102043
[10] Keuthage L, Heider D, Gillespie J W Jr., Gama Haque B Z, Tierney J J, Yarlagadda S, Campbell A and Rinehardt D 2017 Thermoplastic Carbon Fiber Reinforced Body in Structures for Vehicle Crash Application *25th International Technical Conference on the Enhanced Safety of Vehicles (ESV)* **17** 0374
[11] Kießling R, Ihlemann J, Riemer M, Drossel W.-G., A. Dittes, Scharf I, Lampke T, Sharafiev S, Pouya M and Wagner M F-X 2018 A process and load adjusted coating system for metallic inserts in hybrid composites *Prod. Eng. Res. Dev.* https://doi.org/10.1007/s11740-018-0806-3
[12] Kießling R, Landgraf R, Scherzer R and Ihlemann J 2016 Introducing the concept of directly connected rheological elements by reviewing rheological models at large strains *Int. J. Solids Struct.* **97–98** 650–67[13] Throne J L and Beine J 1999 *Thermoformen: Werkstoffe – Verfahren – Anwendung* (München, Wien: Carl Hanser Verlag)
[14] Livermore Software Technology Corporation (LSTC) 2017 *LS-Dyna Keyword User's Manual Volume 2 Material Models LS-Dyna R10.0* (Livermore: Livermore Software Technology Corporation)
[15] Haufe A 2016 Recent developments in reinforced polymer modelling in LS-Dyna *LS-Dyna Forum* Göteburg
[16] Behrens B A, Vucetic M, Neumann A, Osiecki T and Nenad G 2015 Experimental test and FEA of a sheet metal forming process of composite material and steel foil in sandwich design using LS-Dyna *Key Engineering Materials* **651** 439-45
[17] Sachs U, Akkerman R, Haanappel S, Thije R H W and de Rooij M B 2011 Friction in Forming of UD Composites *AIP Conference Proceedings* **1353** 984-89
[18] Haanappel S 2013 Forming of UD fibre reinforced thermoplastics:a critical evaluation of intra-ply shear Enschede *DOI:*10.3990/1.9789036535014
[19] Kießling R and Ihlemann J 2016 Large strain transversely isotropic viscoplasticity by directly connected rheological elements *Proc. Appl. Math. Mech.* **16** 359–60