

Closing the Gap between Process- and Crash Simulation for Composite Materials

Ch. Liebold¹, H. Finckh², T. Günther³, A. Haufe¹

¹ DYNAmore GmbH, Stuttgart, Germany

² Deutsches Institut für Textil- und Faserforschung Denkendorf (DITF), TV, Denkendorf, Germany

³ Volume Graphics GmbH, Heidelberg, Germany

Keywords: mapping, braided tubes, CFRP, computer tomography, pre- and post processing

Abstract

The usage of fiber reinforced plastics (FRP) becomes more and more important in the automotive industry and therefore, the requirements for accurate crash analysis results for such materials increase. As already known from sheet metal forming, process simulations are performed to predict fiber orientations for all kinds of short, long or endless fiber reinforced parts, even though most of the available software tools only use geometrical approximations. On the example of carbon reinforced braided tubes, a mapping tool will be introduced which is able to transfer fiber orientations as well as initial stresses and strains gained from braiding simulations using a high resolution beam mesh onto a coarse crash mesh. The mapping results will be compared with CT-scans taken from such tubes to confirm their accuracy. In the near future, the developed mapping algorithm shall support all kinds of process simulations such as draping or weaving for different element types as well as for different composite materials.

1 Introduction and overview on composite modeling

Composites, especially long fiber reinforced plastic parts are more and more used for light weight constructions in the automotive industry and with the transformation from high performance vehicles such as racing cars to vehicles being produced in high numbers for the regular market, the correct representation of these materials in numerical simulations is more important than ever. So far, LS-DYNA[®] offers more than a dozen of different composite material models which more or less represent the material behaviour of uni-directional (UD) layers or fabric materials. Nevertheless, all of these models require a high degree of homogenization in order to properly represent the material behaviour under crash circumstances.

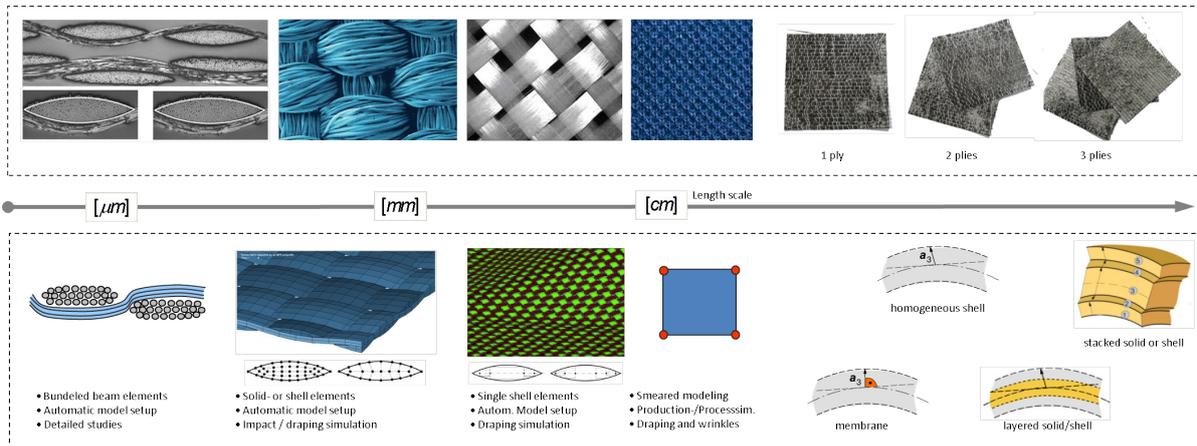


Fig.1: Overview on the length scales being considered when modeling composites [1].

The problems which occur when dealing with such materials shall be illustrated in figure 1. Moving from left to right, it is obvious, that the length scale is changing from micro-meter to a centi-meter scale such as it is used in today's crash simulation. When only considering representative volume elements (RVE), several beam elements can be used to model single fibers in order to understand the micro-mechanical behaviour of the material, but when moving from these highly scientifically driven approaches to the regular crash mesh such as it is shown in the right, one has to make several assumptions to find a proper material representation and quite a few modeling techniques are available which require to simplify the stacking sequence of laminates, for example.

It is well known that the fiber orientation highly influences the material's behaviour of composite materials. Nevertheless, many users of finite element (FE) codes do have to make assumptions about the fiber's orientation when they are meshing their composite part. This is often the case, when the parts are highly curved and no process simulation is performed or measurements from the part processing in general are not available. The resulting meshes represent an idealized model of the complete part and do ignore processing failures such as fiber misalignment, additional thicknesses at the edges of draped parts or areas without any fibers at all. In order to be even more predictive during crash analysis, a first step is to consider such failures and orientations correctly in crushing simulations and therefore, a mapping procedure is necessary to get data from either processing simulations or experimental measurements onto a respective mesh. In this work, a first step into this direction will be taken.

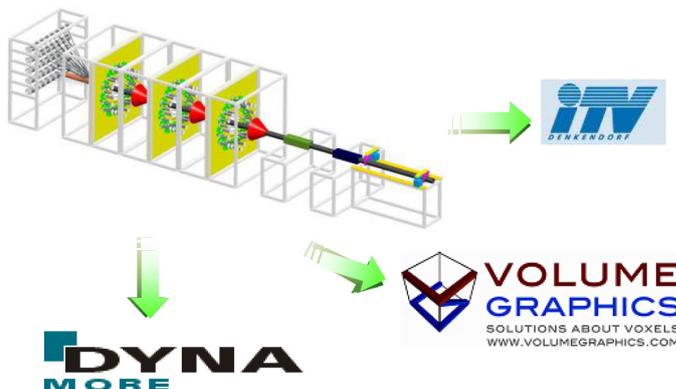


Fig.:2 T-Pult simulation project partners [2]

The presented mapping algorithm is based on a project called T-Pult which is funded by the German ministry of education and research. The aim of the project is to design and build a braiding pultrusion machine for fiber reinforced composites with a thermoplastic resin. The project partners shown in figure 2 are responsible for the experimental measurements with CT-scans and the

numerical simulation and the mapping algorithm for both, the experimental data and results from a processing simulation. Besides the work which is performed for braiding simulations, further modelling techniques which are already available in LS-DYNA[®] are presented in this work and enhancements of the so far developed mapping algorithm will be performed into that direction as well.

2 Process simulation

Besides braiding, resin transfer moulding (RTM), organo sheets, weaving and draping are commonly used technologies for the processing of endless fiber reinforced plastic parts and therefore, these production procedures have to be represented in FE-Simulations in order to capture possible failures during the production and to properly map information about fiber orientation onto a mesh being used in further simulations. This process chain is already quite common for metal forming production processes, not only to predict the producibility of the different parts, but also to transfer the gained information about additional stresses and strains caused by the production process from process simulations onto further meshes used for serviceability simulations.

So far, there are quite a few software solutions on the market, which offer stand-alone simulations for either one of the mentioned production procedures for carbon composite parts. Since some of them are based on geometrical solutions, a short overview on what kind of process simulations can be performed with LS-DYNA[®] will be given. Using FE codes for such analysis has the advantage that besides the information of fiber orientation, also stresses and strains can be used for further calculations.

2.1 Draping

In figure 3, different methods for a draping simulation are shown. On the left side, a simple table cloth is put onto a cylinder and one can see how it wraps around it. These kind of simulations can be performed with *MAT_034 for shell elements and by an in- or decrease of the bending stiffness, different results can be gained. In order to reach a better material representation for such simulations, DYNAMore GmbH works on a new material model which is also capable to output the information of fiber orientations for a further mapping.

The model on the right shows a corner of an L-shaped profile which is modelled with four differently oriented parts represented with beam elements (0°, 90°, 45° -45°). This is another but more time-consuming method which outputs the resulting fiber orientations through the direction of the beam elements and therefore, they can be used directly for a further mapping onto a crash mesh as will be shown later.

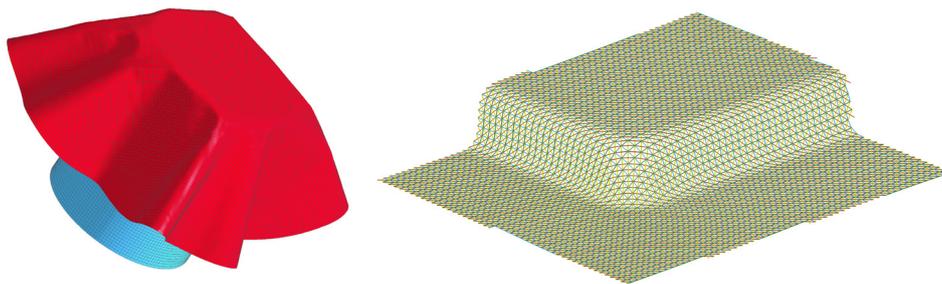


Fig. 3: Draping simulations

2.2 Organo sheets

As shown in figure 4, these kind of simulations are also possible for organo sheets, a fiber-matrix preform which is quite often used in automotive and aerospace industries. The resin is a thermoplastic material. Comparing the fiber orientations marked in white on the real part with the element orientations shown in the figure below, a good match between simulation and the real manufactured part is visible. This model was simulated using shell elements in three layers as a *PART_COMPOSITE. A mapping could be done using the edges of the elements as fiber orientations since so far, the fiber orientation is not given as an output variable in LS-DYNA[®].

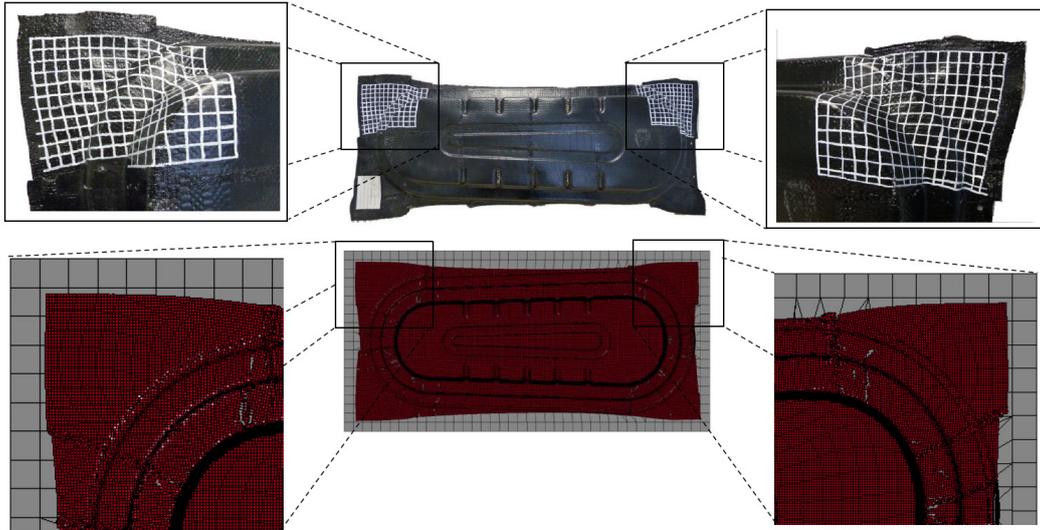


Fig. 4: Simulation of an organo sheet and fiber orientations [1].

2.3 Resin transfer moulding (RTM)

Making the use of the Arbitrary Lagrangian Eulerian (ALE) – method such as it is implemented in LS-DYNA[®], it is also possible to model the transfer moulding process. By varying the porosity which is a function of the packing sequence or the direction of the fibers in the elements, the flow of the resin can be triggered into different directions and therefore a prediction of the resin's flow can be made via a process simulation. When being able to map fiber orientation data from either experimental measurements such as they can be performed with CT-scans or from a draping simulation, it is possible to be even more predictive. The flow through the considered part or porous media can be modeled using *CONSTRAINED_LAGRANGE_IN_SOLID – elements. With the draping simulation results such as they are shown in figure 3 for the L-shaped profile, the simulation shown in figure 5 (left) can be performed. Figure 5 (right) shows how a specified direction with a high porosity can have a strong influence on the simulation results since the resin can only spread out along the horizontal sides of the S-shaped profile.

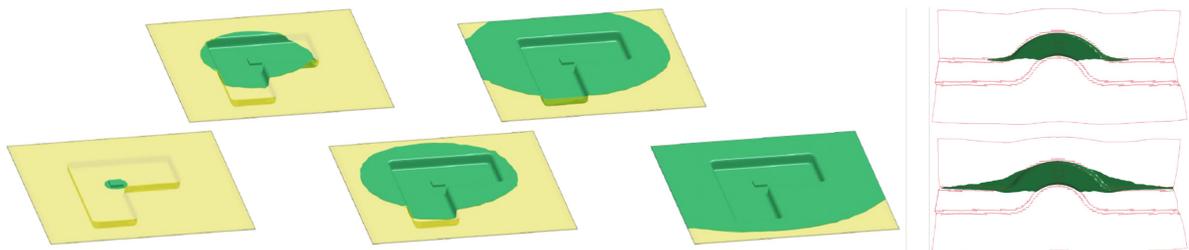


Fig. 5: Examples for a resin transfer moulding for an L-shaped profile (left) and an S-shaped profile (right) [6].

2.4 Weaving

Compared to the simulations presented so far, weaving simulations are rather simple. They can either be performed using single beam elements which makes the effort for the model set up rather complicated, but therefore, full micromechanical behavior can be considered. Such a model is shown in figure 6 (left). The other way would be to use shell- or solid elements for such a simulation such as it is shown in figure 6 (right). This approach is less time consuming for both, the model set up and the actual simulation, but several fibers have to be considered as bundles which reduces the degree of accuracy. Anyways, a mapping can be performed with both, the beam mesh and the solid mesh, even

though for the solid mesh, the element edges have to be considered as output since so far, no LS-DYNA[®] format does consider fiber orientations as history variables.

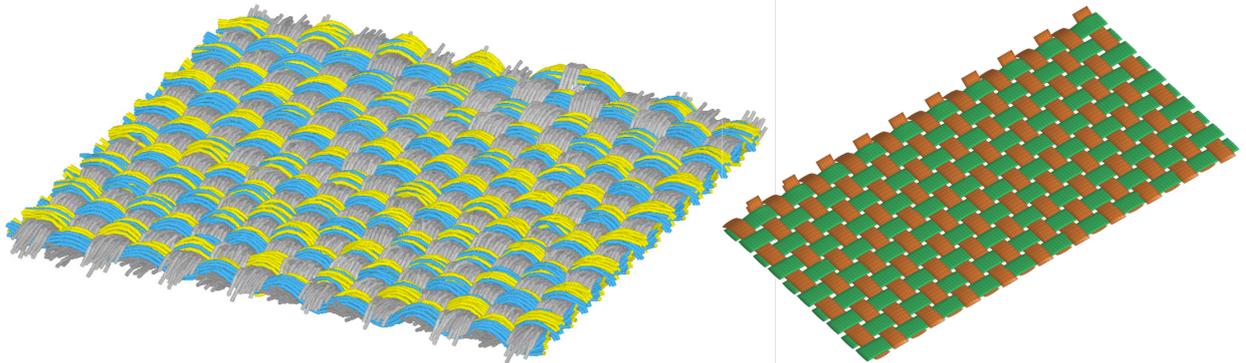


Fig.6.: Weaving simulation with beam elements [2] (left) and solid elements (right).

2.5 Braiding

As it is shown in figure 7, different kinds of braiding simulations can be performed using simple beam elements to represent the fibers: a filament winding procedure (left), a simple braiding (middle) and a braiding simulation considering uni-directional (UD) reinforcement fibers (right). The latter is very time consuming since all the elements have to be modeled from the beginning and a complicated contact algorithm increases the calculation time significantly. Future work shall reduce the calculation time by replacing the elements with a beam-source. Nevertheless, these models can be used for mapping and therefore are sufficient enough for a first approach.

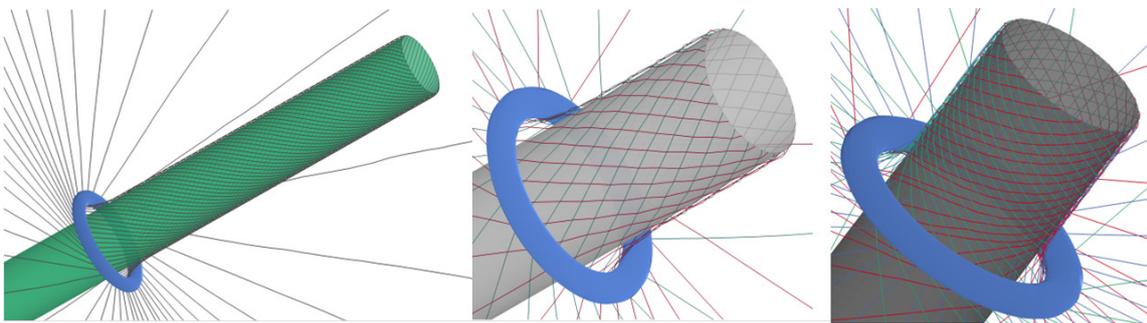


Fig. 7: Filament winding (left), simple braiding (middle) and braiding with uni-directional reinforcement fibers (right).

3 CT - measurements

Besides the braiding simulations, CT-scans are performed in order to get information about the fiber orientation inside the braided tubes. As a first example, a glass fiber reinforced (GFR) tube with three layers is used to get information about the fiber orientation. The advantage of glass fibers is, that they are easier to detect by the CT, even though the project itself will be using carbon fibers. It is known, that the inner- and the outer layers do have different fiber orientation angles, the layer in the middle of the tube contains a thin reinforcement layer which shall be visible in the CT-scans and has to be

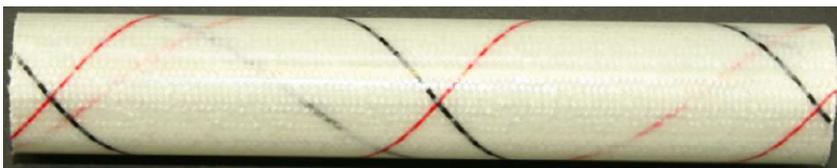


Fig. 8: Glass fiber reinforced tube [3].

produced by Volume Graphics GmbH, Heidelberg, GER.

be considered by a mapping algorithm as well. The tube is shown in figure 8. The measurements are performed at the ITV in Denpendorf, the software which analyzes the measurement is called VGStudio MAX and is

The results of such measurements do contain a fiber orientation tensor and the fiber volume fraction of a given volume element. They are mapped onto three different mesh sizes such as they are given in table 1. The meshes considered so far are all tetra-element meshes. The number of layers indicated in table 1 refers to the number of elements through the thickness of one real layer in the GFR – tube. The mesh size is definitely not relevant for crash simulations, but it gives a first overview on how these data can be mapped and visualized with a LS-DYNA[®] mesh. The mapping results and further mapping strategies will be presented later on.

	<u>Num. of Elements</u>	<u>Layers</u>	<u>Element-Size</u>
Coarse Mesh	38420	1	0,3 - 1,25 mm
Mean Mesh	150039	2	0,15 - 0,62 mm
Fine Mesh	15078400	4	0,075 - 0,1 mm

Tab. 1: Information about the meshes used for a mapping from CT-scans.

In this first investigational approach, the output of the fiber orientation tensor which can be gained by:

$$a_{ij} = \frac{1}{n} \sum_{k=1}^n a_{ij}^k = \frac{1}{n} \left(\sum_{k=1}^n p_i^k p_j^k \right) = \begin{pmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{pmatrix} \quad (1)$$

with:

$$p = \begin{pmatrix} \sin \theta \cos \phi \\ \sin \theta \sin \phi \\ \cos \theta \end{pmatrix} \quad (2)$$

as the normalized vector of the fiber orientations in polar coordinates [4] is mapped onto one of the given tetra element meshes with the *INITIAL_STRESS_SOLID – card representing the entry of the six different parameters of the fiber orientation tensor. The results of such CT-scans are shown in figure 9.

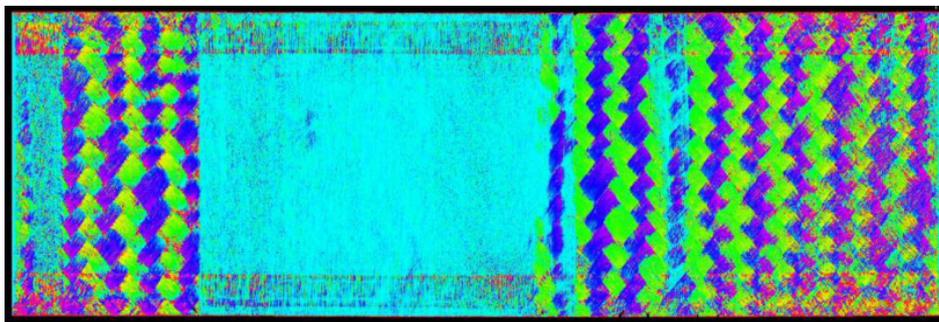


Fig. 9: Layer 1 (upper) and layer 2 with UD-reinforcement fibers (lower) of the GFR tube.

4 Homogenization strategies

As already mentioned, different strategies to consider fiber orientations in a crash simulation are available in LS-DYNA®. Figure 10 [5] which is taken from the user's manual illustrates the number of possibilities being available for the different versions of AOPT which can be defined in the material card. The highlighted AOPT options show the ones which are the most relevant and therefore are used for the implementation of the mapping algorithm. Using AOPT = 0 can be recommended since this model does align the main fiber orientation along the element's main axis. Nevertheless, this model is the hardest to implement since orientations may vary from element to element. For composite structures with a high degree of curvatures, AOPT = 2 cannot be recommended since it defines the main material direction by a vector given in the material card. In case that the material direction varies in each element which is often the case for draped parts for example, a high number of material cards has to be defined which increases the calculation time for such a model. The usage of AOPT = 3 can be recommended for tube-like structures such as they are considered in the T-Pult project. Therefore, this feature will be implemented in the presented mapping algorithm as well.

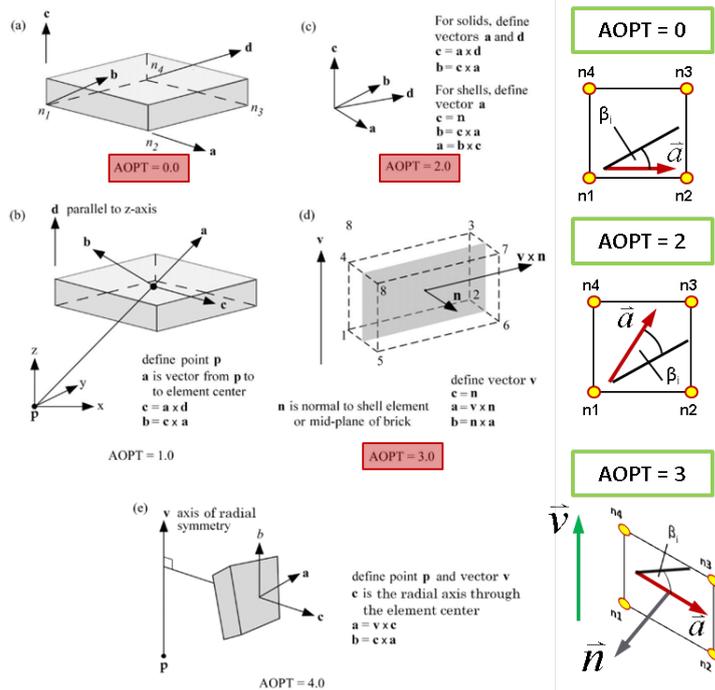


Fig. 10: Available AOPT parameter for fiber orientation angle β [5].

defined which increases the calculation time for such a model. The usage of AOPT = 3 can be recommended for tube-like structures such as they are considered in the T-Pult project. Therefore, this feature will be implemented in the presented mapping algorithm as well.

Another challenge for the implementation of a mapping algorithm has to be the inclusion of various homogenization strategies. Going from process simulation to a regular crash mesh might also include a reduction of layers which are modelled with shell or solid elements or a reduction of integration points. In figure 11, this problem is illustrated for the example shown in figure 3. The orientations of four parts being represented by beam elements shall be mapped onto a mesh being relevant for crash simulations. In this case, the beams represent four different fiber orientations in one element, but they could also be a layered structure with one main fiber direction for each element or a *PART_ or *ELEMENT_SHELL_COMPOSITE. As figure 11 shows, it is possible to either map the different orientations onto several layers (e.g. one layer for each part) or to map them onto integration points of one *PART_ or *ELEMENT_SHELL_COMPOSITE. The last option is usually more relevant for industrial applications and is therefore recommended. When using *ELEMENT_SHELL_COMPOSITE, the orientations can be different for each integration point in each element and a mapping together with AOPT = 0 is a good choice for this option.

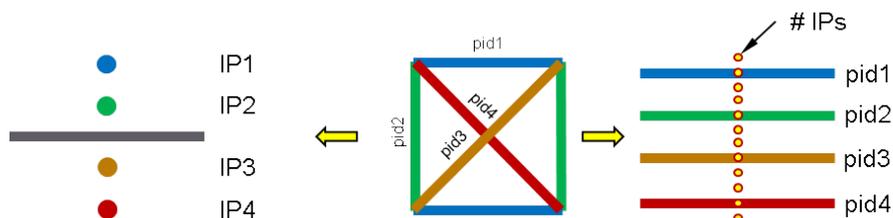


Fig. 11: Homogenization strategies for fiber orientations from process simulations.

5 Mapping results

5.1 Mapping of experimental data

As already mentioned, a part of the T-Pult project is to map fiber orientations from CT – measurements onto a finite element mesh. In a first step, fiber orientation tensors are written onto *INITIAL_STRESS_SOLID – cards for visualization purposes only. The results are given in figure 12. It is obvious, that smaller tetra elements do represent the different orientations much better than the elements used in the coarse mesh. The visible UD-reinforcement layer shown in figure 9 is getting visible for the mean mesh size and can be clearly identified for the fine mesh. For the fine mesh, the

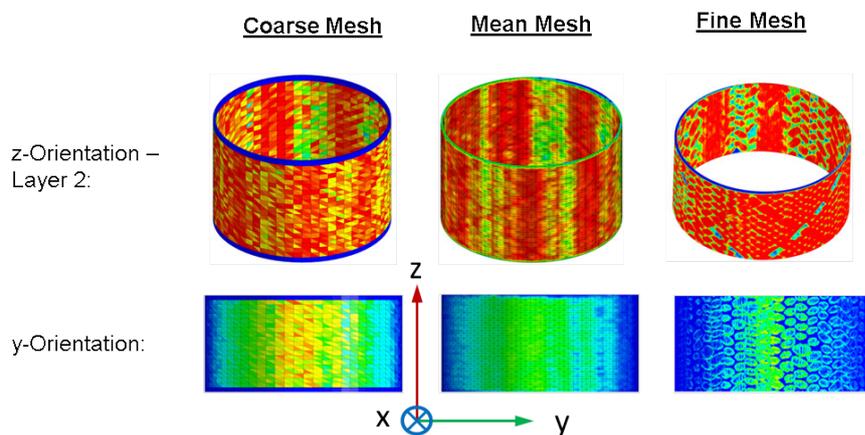


Fig. 12: Mapping results from experimental measurements.

orientations have to be taken since the orientation tensor simply represents the average orientation inside a defined volume – a next step will be to map the main fiber orientations which can be identified by VGStudio MAX onto a more crash relevant structure.

braided structures of GFR – tube can be depicted clearly. The orientation also seems to be represented quite well. From the point of view which is shown in the lower series of the results, the tensor must have a high number of elements being oriented in y-direction, as well as in z-direction since the fibers wrap around the tube. Nevertheless, further steps towards a better output of the

5.2 Mapping of simulation results

In order to validate the mapping algorithm, the results for a mapping with the different homogenization strategies and the different options for AOPT are shown. Figure 13 illustrates the results which were

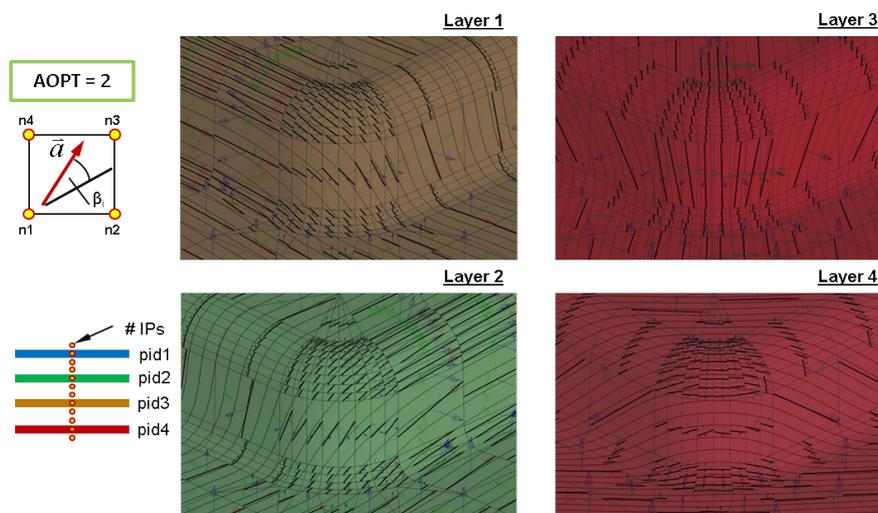


Fig. 13: Mapping onto different layers for each orientation with AOPT = 2

gained for the beam mesh given in figure 3 using AOPT = 2. This mapping is quite simple since the vector of the fiber orientation can be a direct input into the material card but as mentioned before, many material cards have to be generated and therefore, a following crash simulation with such a structure is not realistic. Nevertheless, these results are a first proof of concept and the algorithm can be improved from this basic state.

The more relevant case which is a mapping onto a mesh which does define the main material direction via the orientation of the elements itself is shown in figure 14. The results are similar to the ones already shown since the beam mesh given in figure 3 is used again for this first benchmark. Instead of mapping each orientation onto one layer as shown before, the *ELEMENT_SHELL_COMPOSITE – card is introduced to define the fiber orientation of each layer with the β_i angle independent from the element orientation using AOPT = 0 in the material card.

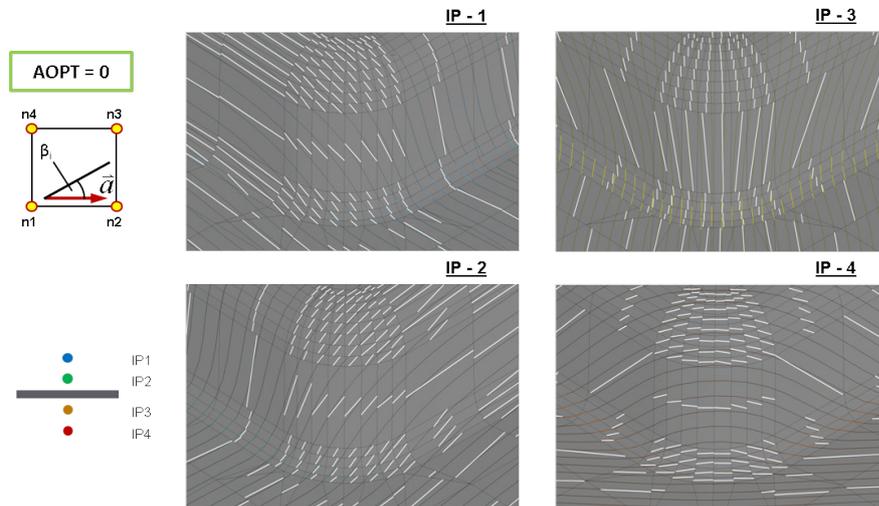


Fig. 14: Mapping onto different integration points with AOPT = 0.

for each of the two main directions onto a mesh using *ELEMENT_SHELL_COMPOSITE elements to define the lay up per integration point. Also conceivable but not yet implemented is a mapping onto a homogenized material which covers the two main directions of such a braided tube like a material model for fabrics such as *MAT_LAMINATED_COMPOSITE_FABRIC (*MAT_058) would do. Nevertheless, an implementation of such a feature can be done easily since the way of defining the respective fiber orientations is already known.

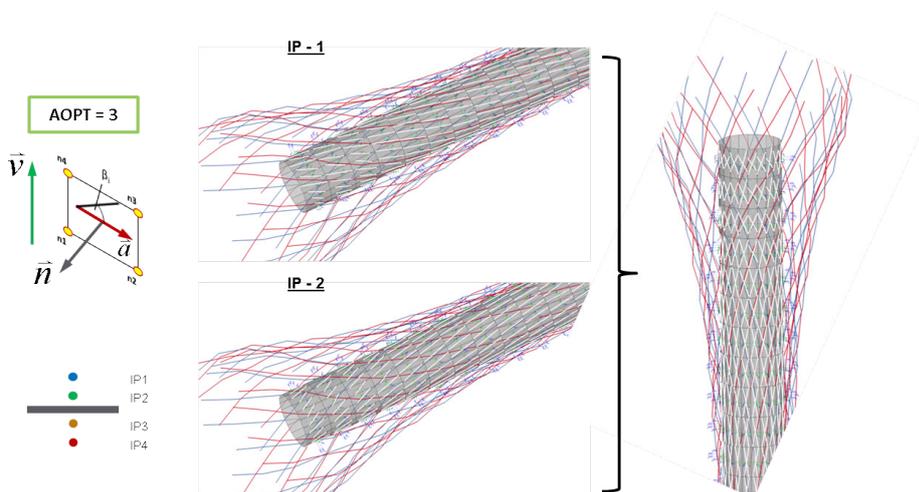


Fig. 15: Mapping onto different integration points with AOPT = 3.

writes such vector files from a beam element mesh is already available in the presented mapping algorithm. Main drawback for this option is, that the orientations of the different parts have to be written into different vector files which will be added up to plies by the user in the new EleTol → EleEdit → Composite environment in LS-PrePost[®]. The output will be independent from the elements orientation (AOPT = 0) and *ELEMENT_SHELL_COMPOSITE – elements are used to define the different plies and their orientations.

The case which might also be relevant for the braided tubes being produced for the T-Pult project is AOPT = 3, defining the vector v along the tube's main axis. Figure 15 shows the mapping results for a braided structure which is similar to the one shown in the middle of figure 7.

Enhancements towards a mapping of more complicated braided tubes can be done easily. Again, the mapping is performed

In order to complete this listing of mapping solutions it should be mentioned that LS-PrePost[®] does offer such a feature as well. It can map information from vector files onto the actual loaded mesh. This method is available since release 4.1. Therefore, a feature which

6 Conclusion and outlook

To be able to bridge the gap between process- and crash simulation, a mapping tool was presented which is able to use most of the options available in LS-DYNA® to define the main material orientations for each element. The development is driven by the T-Pult project which is funded by the German ministry of education and research and therefore, the main focus of the presented algorithm is to offer a mapping solution for tube-like structures for both, simulations and experimental measurements.

As already mentioned, the mapping from CT-scans is just at the beginning and further improvements will be made to get better results for this application. Main focus thereby will be on the mapping of the real fiber orientations and not just the orientation tensor which is the mean of the orientations being found in a defined volume.

The mapping feature which performs a transformation from fiber orientations gained through numerical simulations onto a crash mesh is working well in case that the mapping is performed from a beam mesh onto a shell mesh either with several layers or with the *ELEMENT_SHELL_COMPOSITE – option, which is of course more relevant for the automotive industry. Further improvements have to be taken in order to consider certain material models as well as to reduce the number of layers which are mapped from one mesh to another. An additional feature which will be implemented in the near future is the mapping of fiber orientations which are an output of the material model itself. Therefore, additional history variables or another output file has to be generated which contains information about the fiber orientation compared to the local element coordinate system. First steps into that direction were already performed at the DYNAMore GmbH for a *USER_DEFINED_MATERIAL_MODEL which characterizes the behavior of endless fiber reinforced composite materials during a draping process.

Another goal is to get further information about the influence of the fiber orientation and packing sequence on the porosity of a laminate. Knowing that, different porosities can be mapped onto the elements of a mesh which is used for a RTM process simulation such as it was explained in section 2.3. With all these measures taken, an evaluation process can be launched which shall proof the benefits of including the process simulation into the composite part design and development. A higher degree of predictability and a more realistic numerical model compared to reality can be expected.

7 Literature

- [1] Haufe, A.: "Die Simulationsprozesskette für faserbasierte Werkstoffe: Von der Herstellungssimulation zu Aussagen der Gebrauchstauglichkeit", conference proceedings, Fachkongress Composite Simulation, Fellbach, GER, 2013
- [2] Finckh, H. et. Al.: "Prozesssimulation am ITV – Möglichkeiten für Faserverbundstrukturen", conference proceedings, Fachkongress Composite Simulation, Ludwigsburg, GER, 2012
- [3] Finckh, H., Günther, T.: High resolution fibre orientation analyses of carbon and glass fibre composites, conference proceedings, 8th GE X-ray Forum 2012, Hamburg, GER, 2012
- [4] Kastner, J. et. Al.: „Pipeline zur dreidimensionalen Auswertung und Visualisierung der Faserverteilung in glasfaserverstärkten Kunststoffteilen aus μ -Röntgen-Computertomografiedaten“, conference proceedings, DACH-Jahrestagung, St. Gallen, CH, 2008
- [5] Livermore Software Technology Corporation (LSTC): „LS-DYNA Keyword User’s Manual – Vol. II – Material Models“, Livermore, CA, USA, 2012
- [6] Kloeppe, T. et. al.: "Möglichkeiten zur Abbildung der Prozesskette von FVK in LS-DYNA“, conference proceedings, Leichtbau-WM, Schladming, AT, 2013

Acknowledgment

This research and development project T-Pult is funded by the German Federal Ministry of Education and Research (BMBF) within the "Framework Concept "Research for Tomorrow's Production " (02PJ2180) and managed by the Project Management Agency Karlsruhe (PTKA). The author is responsible for the contents of this publication."