Mapping IM Process Information: Use of ANSA & Material Model MFGenYld+CrachFEM

Gabriele Fruhmann[1], Alexander Rager[1], Gernot Oberhofer[2], Michael Richter[2],
Michael Tryfonidis[3], Panagiotis Fotopoulos[3], Philipp Mäser[4], Sebastian Mönnich[5]

[1] BMW Group AG
[2] MATFEM Ingenieurgesellschaft mbH
[3] BETA CAE Systems SA
[4] ARRK Engineering GmbH
[5] PEG - Plastics Engineering Group GmbH

Abstract

Injection molded (IM) parts show local anisotropic behavior. If results of IM simulations are given, mapping of local fiber orientations/weld lines can be introduced. Request: The material model can approximate the material in an early stage of development without process data as a homogenized isotropic material and later in a more detailed stage by using process simulation results with local properties in the part.

It will be shown that this is possible with the material model MF GenYld+CrachFEM in combination with pre-processing in ANSA and IM results from 3D Moldflow® simulation. This information is mapped with the ANSA mapping tool onto the corresponding mesh for crash simulation. Because of the different meshes in Moldflow® and ANSYS® LS-DYNA®, settings for the mapping and careful handling of the generated weld lines are of high relevance.

1 Motivation and Framework

In today's vehicles, the number of components made of plastic is increasing in both structural and functional components that are relevant for meeting crash requirements. The size of vehicles is also growing, leading to ever larger and more complex injection molded components. In order to be able to manufacture these efficiently, the demands on injection molding processes and tools are increasing and the integrative design of the production process and crash protection are becoming more and more important. In parallel, the possibilities in process simulation as well as in FE crash simulation are expanding and the development of the digital twin is progressing. This means that more design steps and validations can be carried out with the help of digital tools.

In the frame of this article, the procedure for the transmission of the relevant results from the injection molding simulation to the FE crash simulation for the following program environments is shown:

- Moldflow® for process simulation with 3D mesh (tetrahedrons)
- ANSA as a pre-processor for the preparation of the input decks for ANSYS® LS-DYNA®
- ANSYS® LS-DYNA® for FE crash simulation with 2D shell mesh
- User material model MF GenYld+CrachFEM coupled with ANSYS® LS-DYNA® for the description of the material properties

This process chain was developed together with our development partners BETA CAE Systems SA (now part of Cadence Design Systems, Inc.), MATFEM Ingenieurgesellschaft mbH, ARRK Engineering GmbH and PEG - Plastics Engineering Group GmbH and has now been successfully implemented at the BMW Group AG in series development for selected components.

2 Process Overview

The process chain consists of the process simulation in Moldflow® - usually carried out by our suppliers, the mapping of the relevant information from the process simulation to the FE component in the crash simulation with the help of the pre-processor ANSA using the tools “Results Mapper”, “Weld Lines” and “Results to CrachFEM” as well as the explicit solver ANSYS® LS-DYNA® in extension with the user material model MF GenYld+CrachFEM, see Fig. 1. Animator is preferred for post-processing and META is used in special cases.

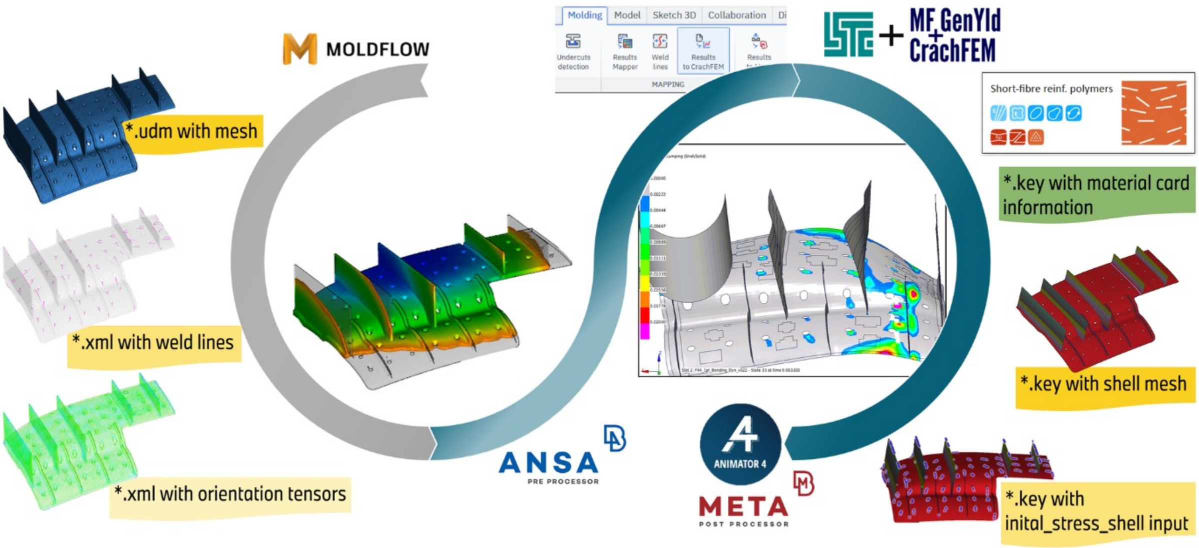


Fig.1: Process chain.

Compared to conventional modeling of injection molded components with homogeneous isotropic material properties, this more complex procedure is only used at BMW for selected components. The focus is primarily on structural components with high crash requirements, such as the air flap control system [1] and specific frames with high structural functionality.

The information from the process simulation that is currently deemed as relevant for mapping are the weld lines for unreinforced, filled and short and long fiber-reinforced injection molded components and in addition, the local orientation of the fibers for short and long fiber-reinforced components. This usually results in two to three files that must be provided as input for the mapping in ANSA, see left hand side in Fig. 1.

For mapping in ANSA, the tetrahedron mesh from the process simulation and midsurface FE shell mesh for crash simulation must first be positioned relative to one another. Then the weld lines and, if necessary, the fiber orientations are mapped to the elements and corresponding integration points. All mapped information is summarized under the keyword **`*INITIAL_STRESS_SHELL`**, considering the specific history variable setup of MF GenYld+CrachFEM for the unreinforced or short and long fiber reinforced polymer materials, Fig. 1 right hand side.

The information for **`*INITIAL_STRESS_SHELL`** is exported to a separate *.key file for the respective component. By including this additional ASCII-file containing the mapping information, flexible and easy control of the accuracy of the material modeling in the crash simulation is enabled. This means that an MF GenYld+CrachFEM material card can already be used to simulate in early development phases in which no information from the process simulation is available, as it automatically detects whether mapping information is available or not during initialization. If mapping information is missing, the material model internally turns on isotropic material behavior and thus reflects a less detailed material behavior.

During the development process, the information from the mapping can be added to the model without changing the material card and thus retaining same material ID (MID). If the mapping is recognized for a component, each integration point is initialized with the information from the process simulation and this information is then processed via MF GenYld+CrachFEM for a more detailed reflection of the local material properties.

The results of the mapping as well as of the entire simulation can be visualized via the postprocessors Animator or META.

3 Influence of the Process on the Material Properties

A closer look at the design of injection molded components shows that it is generally a well-coordinated chain with many individual steps and boundary conditions. High quality injection molded components can only be achieved if the material, injection molding process and geometry of the component are tailored to the application [2] - and above all to the loads and boundary conditions, Fig. 2.

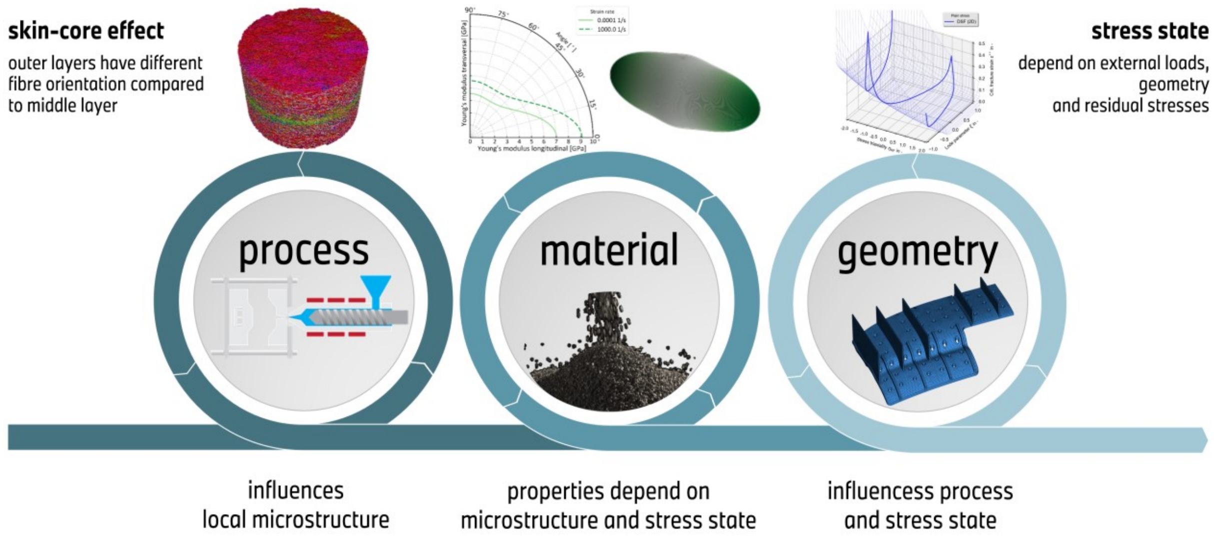


Fig.2: Interaction of process, material and geometry, taking into account the component loads.

Based on the component requirements, a material is determined that is suitable for the injection molding process. The component geometry and material selection require compliance with certain areas of the process parameters in order to be able to produce a defect-free component. The interaction of process control with the associated process parameters, the material with its rheological and thermal properties and the framework conditions created by the geometry influence the formation of the local microstructure in the material [2-6]. This microstructure in turn, together with the stress states caused by the outer loads in the component, determines the behavior of the material.

A simple example of an injection-molded tension bar can be used to show the differences that can result from the positioning of sprue points with regard to the microstructure, especially in the case of short or long fiber-reinforced materials, see Fig. 3.

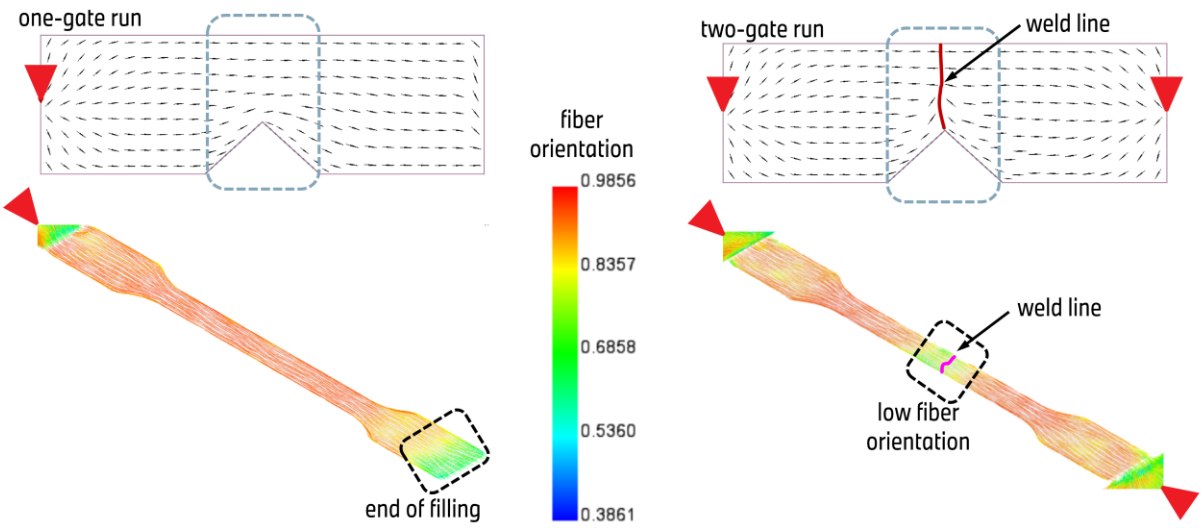


Fig.3: Influence of the sprue position of a tension rod on fiber orientation and position of the weld line.

Using a one-gate run for injection of the tensile bar (Fig 3 left), no weld line is formed and in the middle area there is a high orientation of the fibers in the longitudinal direction of the tension bar. If the mechanical properties of the bar are determined under unidirectional load in the longitudinal direction, e. g. according to DIN EN ISO 527-1/2, the result shows comparatively high stiffness and strength with low elongation at yield and at break compared to measurements with fiber orientations transverse to the direction of load.

If the tensile bar is manufactured with the same test specimen geometry with two sprues, a weld line forms in the middle area, which leads to a significantly lower fiber orientation in the middle area compared to the tensile bar with one sprue. A comparable unidirectional test of this tensile bar will show a significantly lower strength with even less ductility, depending on the choice of process parameters, compare Fig. 4 on the left.

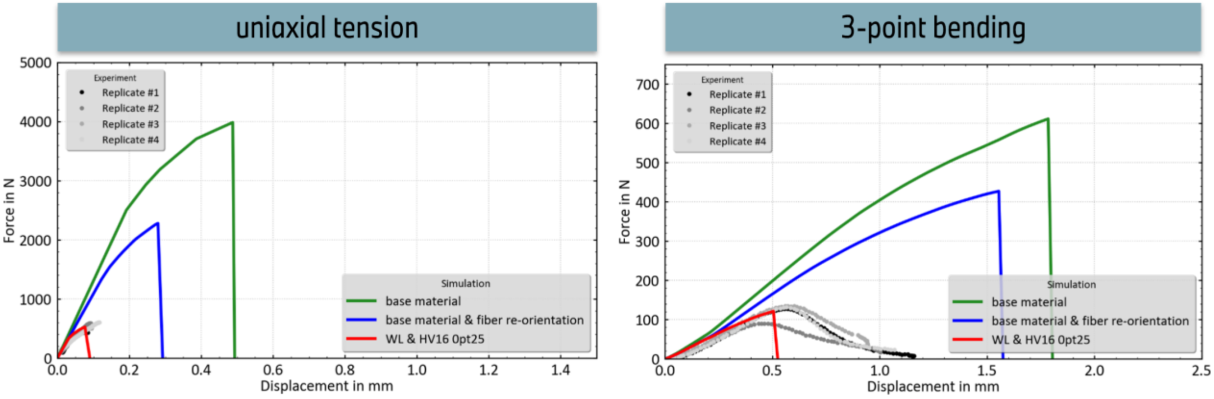


Fig.4: *Comparison of mechanical properties under tension and bending load with/without weld line. green: material properties with fiber orientation Fig. 3 left; blue: material properties with fiber orientation Fig. 3 right but without scaling for weld line; red: material properties with fiber orientation Fig. 3 right and with scaling for weld line.*

Now the middle section of the tension bar can also be subjected to a 3-point bending test in accordance with DIN EN ISO 178 - this corresponds to a changed stress state. A comparison of the two tension bars regarding their bending behavior shows a striking difference in the bending strength and the achievable deflection, see Fig. 4 on the right.

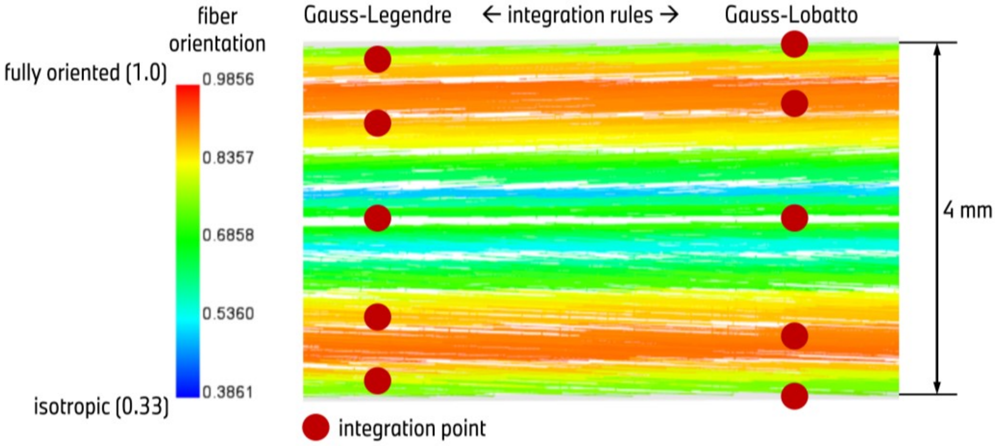


Fig.5: *Result of a process simulation for the fiber orientation in a plate with a thickness of 4 mm. Possible layers for integration points across the thickness of shell elements are also shown.*

In addition to weld lines with its different characteristics, the skin-core effect develops depending on the material, the local flow conditions and the local existing wall thickness. This refers to the different orientation between the surface layers and the middle layer. A good process simulation (Fig. 5) depicts this effect, and it can be considered in the mapping.
As the simple example shows, very different local microstructures can be present, which, in interaction with outer loads and the resulting local stress states, lead to very different material behavior.
The target of the design is the clever combination of microstructures that form through the considered positioning of sprue points, geometric progressions and process parameters, for the most robust behavior possible with regard to the loads that occur and functional requirements.

4 Mapping Challenges and Details

Successful mapping requires that the process simulation with

- the corresponding geometry to the FE component in the crash simulation with
- a mesh (3D tetrahedron) that is sufficiently accurate for the geometry,
- the appropriate material model and parameters for the rheological and thermal behavior and
- a sufficiently accurately modeled gating system
- and the correct process parameters

is carried out in Moldflow®. Our experience shows that mapping only makes sense once the sprue points, the material and a rough process setup have been defined, and the first iterations deliver an error-free injection molding process.
As artifacts often occur in Moldflow® when determining the weld lines, i. e. weld lines are shown that do not actually exist, an additional macro was programmed for Moldflow® that brings together several sources of information within Moldflow® - e. g. the "Weld Lines" with the "Weld surface formation" and performs plausibility checks. This allows many artifacts to be eliminated and supports faster and higher quality mapping of the weld lines in ANSA, see Fig. 6.

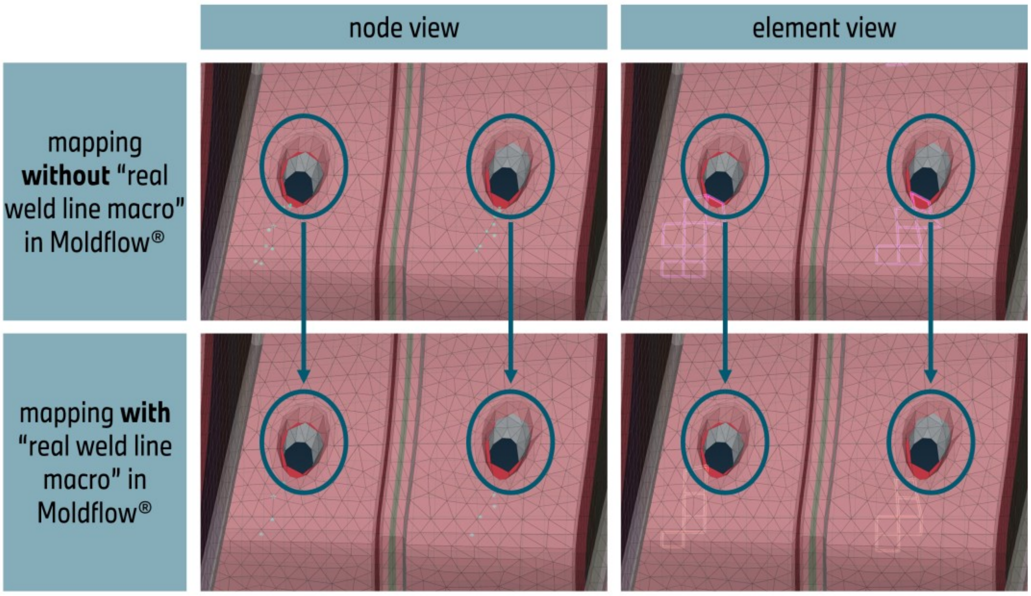
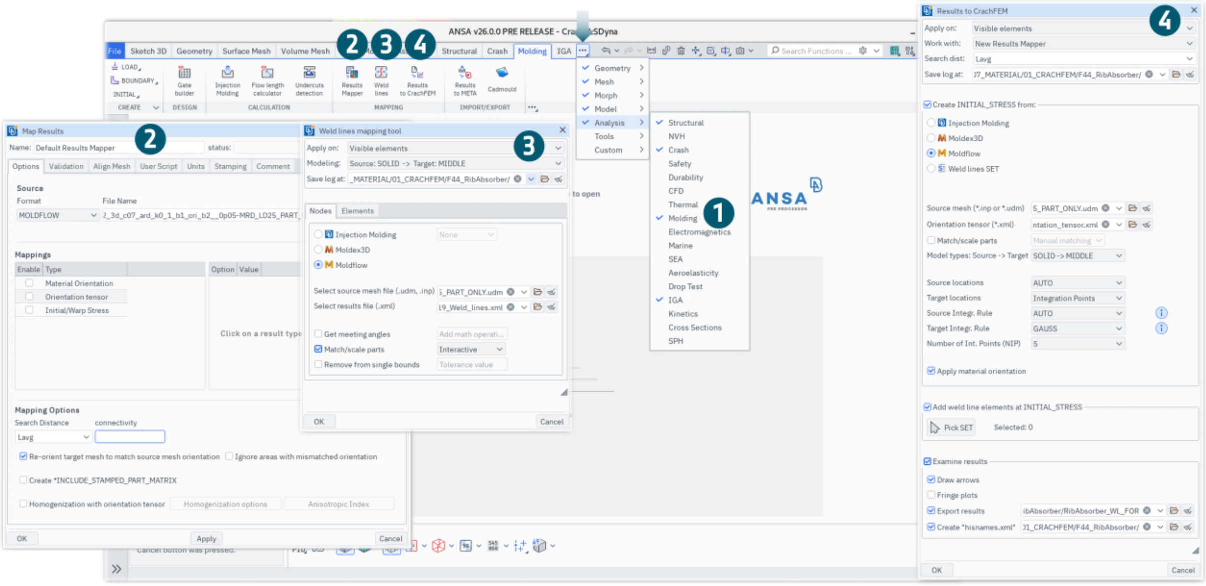


Fig.6: Comparison of mapping behavior without and with "real weld line macro" in Moldflow® shown in ANSA with semitransparent tetrahedron mesh of the process simulation together with the shell mesh (red) for crash simulation.

The results now available from the process simulation are prepared and exported for the ANSYS® LS-DYNA® input deck with the help of individual tools in the ANSA preprocessor (Fig. 7). Typically, the transformation matrix (Fig. 8 below) is generated with "Map Results" (2), the weld lines are mapped with the "Weld line mapping Tool" (3) and the fiber orientations are mapped with "Results to CrachFEM" (4) and, if necessary, visualized and exported as an input deck. To check the mapping, additional files can be exported for visualization in META (4, bottom).



Map Results
Name: Default Results Mapper
status:
Source
Format: MOLDFLOW
File Name: 2_3d_c07_ard_k0_1_b1_ori_b2__0p05-MRD_LD25_PART_
Mappings
Enable	Type
☐	Material Orientation
☐	Orientation tensor
☐	Initial/Warp Stress

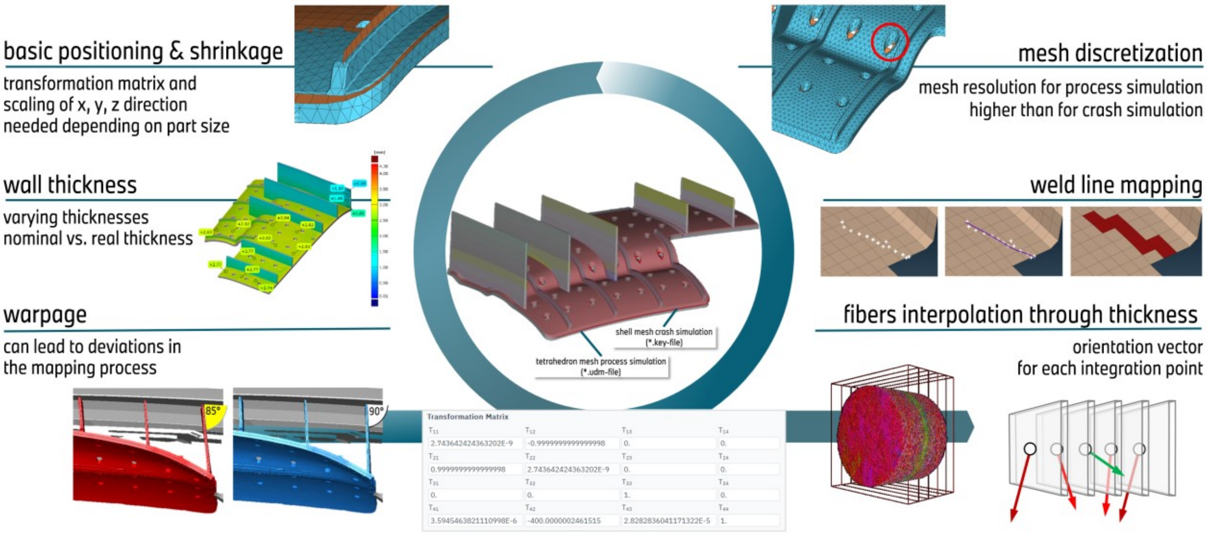
Option | Value
Click on a result type
Mapping Options
Search Distance: Lavg
connectivity
☑ Re-orient target mesh to match source mesh orientation ☐ Ignore areas with mismatched orientation
☐ Create *INCLUDE_STAMPED_PART_MATRIX
☐ Homogenization with orientation tensor   Homogenization options   Anisotropic Index
Cancel button was pressed.

Weld lines mapping tool
Apply on: Visible elements
Modeling: Source: SOLID -> Target: MIDDLE
Save log at: _MATERIAL/01_CRACHFEM/F44_RibAbsorber/
○ Injection Molding   None
○ Moldex3D
● Moldflow
Select source mesh file (.udm, .inp): 5_PART_ONLY.udm
Select results file (.xml): 19_Weld_lines.xml
☐ Get meeting angles   Add math operati...
☑ Match/scale parts   Interactive
☐ Remove from single bounds   Tolerance value

Results to CrachFEM
Apply on: Visible elements
Work with: New Results Mapper
Search dist: Lavg
Save log at: 07_MATERIAL/01_CRACHFEM/F44_RibAbsorber/
☑ Create INITIAL_STRESS from:
○ Injection Molding
○ Moldex3D
● Moldflow
○ Weld lines SET
Source mesh (*.inp or *.udm): 5_PART_ONLY.udm
Orientation tensor (*.xml): ntation_tensor.xml
☐ Match/scale parts   Manual matching
Model types: Source -> Target: SOLID -> MIDDLE
Source locations: AUTO
Target locations: Integration Points
Source Integr. Rule: AUTO
Target Integr. Rule: GAUSS
Number of Int. Points (NIP): 5
☑ Apply material orientation
☑ Add weld line elements at INITIAL_STRESS
Selected: 0
☑ Examine results
☑ Draw arrows
☐ Fringe plots
☑ Export results: ibAbsorber/RibAbsorber_WL_FOR
☑ Create "hisnames.xml": 01_CRACHFEM/F44_RibAbsorber/

Fig.7: ANSA mapping tools.

The first step in mapping is positioning the two different FE meshes relative to one another, see Fig. 8. It is important to remember that shrinkage and distortion must be considered, especially for large components. ANSA offers automatic positioning and scaling, Fig. 7 (2-4), which can be readjusted manually if necessary. It is always advisable to check the positioning visually or automatically at relevant points. The result is stored in a so-called “results mapper” and can be reused.



Transformation Matrix

T_{11}	T_{12}	T_{13}	T_{14}
2.743642424363202E-9	-0.9999999999999998	0.	0.
T_{21}	T_{22}	T_{23}	T_{24}
0.9999999999999998	2.743642424363202E-9	0.	0.
T_{31}	T_{32}	T_{33}	T_{34}
0.	0.	1.	0.
T_{41}	T_{42}	T_{43}	T_{44}
3.5945463821110998E-6	-400.0000002461515	2.8282836041171322E-5	1.

Fig.8: Steps in mapping and their challenges.

Depending on the mesh size in relation to the geometric details, unavoidable deviations may occur in different areas, compare Fig. 6. The mapping algorithm must deliver robust results for these deviations. In addition, due to tight schedules, there may be - usually small - differences between the geometry for the injection molding simulation and the geometry for the crash simulation. These deviations must provide reproducible results during mapping, i. e. no initialization values should be generated that lead to a deviation from the isotropic material state.

Further deviations can occur with wall thicknesses in the area of the mold parting surface. If this plane is relevant with regard to bending loads, the actual wall thickness must be carefully checked, Fig 8 left.

As soon as the positioning is complete, the weld lines, Fig. 7 (3), can be mapped in the second step. To do this, the *.udm file and the *.xml file with the weld line information are read in and, with the help of the "results mapper" already created, a node set is generated correctly positioned in relation to the shell geometry. This node set can be visualized in ANSA GUI and manually manipulated. In the following step the nodes are clustered and approximated using spline functions. In the spline set, it is possible to use either all splines or a subset for the consecutive projection onto the mid-surface shell mesh. All elements of the shell mesh onto which a node has been projected are summarized in a set named “Weld Lines Shell Elements”. This set can also be visualized for checking purposes, see Fig. 6 and Fig. 8 on the right. If artifacts in the Moldflow® simulation result in broad weld lines, it is advisable to reduce these to a realistic width by deleting individual elements so that any strain localizations that may occur later under load are not overlooked accidently. The influence of weld lines on mechanical properties is shown in Fig. 4.

For fiber-reinforced materials, the third step involves mapping the fiber orientations, Fig. 7 (4) and Fig. 8 bottom right, using the *.udm file and the *.xml file with the fiber orientation information from the "Fiber orientation tensor" from Moldflow®. To do this, the mapping algorithm must correctly determine and average the tensors for the fiber orientations regarding the integration points of the shell mesh. The position of the integration points of the source mesh and the target mesh is essential for this. Additional information about the integration points is therefore required, e. g. by selecting the integration rule, see Fig. 5 and Fig. 7 (4). Wall thickness and the number of integration points over the thickness can - as shown in Fig. 5 - strongly influence the averaged tensor. As can be seen in Fig. 9, even slight deviations in the thickness direction can lead to very different mechanical properties.

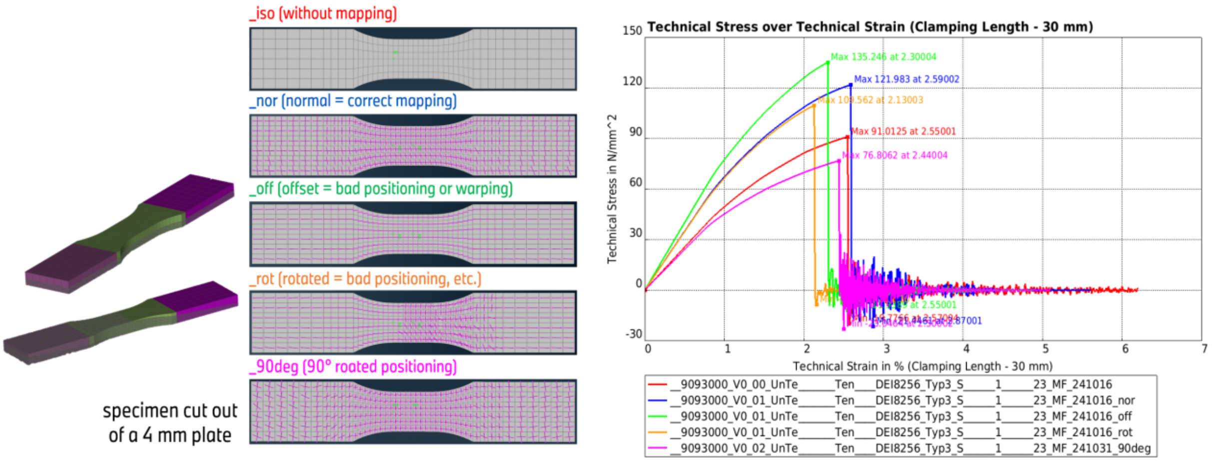


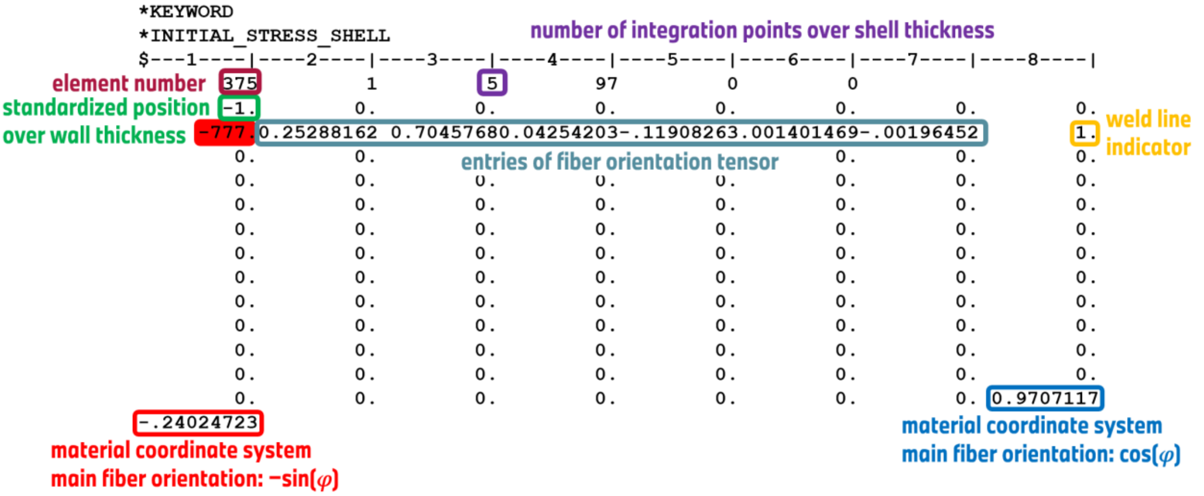
Fig.9: Change in mechanical properties due to deviations in mapped fiber orientations shown in ANSA.

After successful mapping, the result is generated for the keyword **`*INITIAL_STRESS_SHELL`** by ANSA, considering the use of the user material model MF GenYld+CrachFEM. The content for each individual component is exported to a separate *.key file for further use in ANSYS® LS-DYNA® crash simulations. This makes it easy to create input decks for simulations both with and without mapping.

The use of MF GenYld+CrachFEM defines the data in **`*INITIAL_STRESS_SHELL`**. Fig. 10 shows an excerpt of the assignment for an integration point with 97 history variables. For each element, in relation to the material coordinate system and the respective integration point, which is defined by the normalized wall thickness, the information

- about the material sub model to be used (-777),
- the averaged orientation tensor,
- the weld line, if present, and
- the position of the main direction of the tensor

is transferred for initialization.

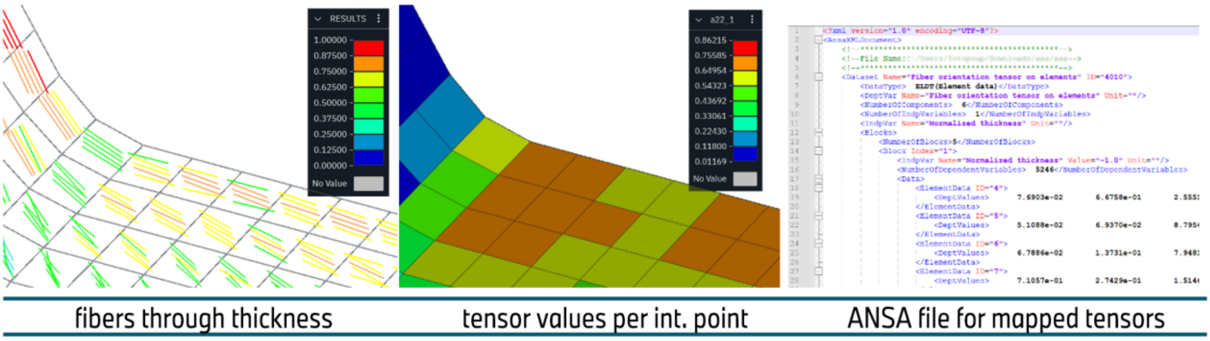


```
*KEYWORD
*INITIAL_STRESS_SHELL
$---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|
      375        1         5        97         0         0
      -1.        0.        0.        0.        0.        0.        0.        0.
    -777.0.25288162 0.70457680.04254203-.11908263.001401469-.00196452        1.
       0.        0.                                      0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.        0.
       0.        0.        0.        0.        0.        0.        0.0.9707117
-.24024723
```

*Fig.10: Dedicated data format in **`*INITIAL_STRESS_SHELL`** for MF GenYld+CrachFEM material model with the information from the process simulation for the initialization of the FE component in the crash simulation.*

Due to the high complexity and amount of information, good visualization concepts are required to check the mapped results. In the case of weld lines, particular attention is paid to the width and the start and end areas. In the case of fiber orientations, the correct orientation at high degrees of orientation and the course across the wall thickness must be checked in particular. From version V25.1 onwards ANSA now offers good visualization options for early checks, like

- lines representing the fibers per element per integration point, e. g. Fig. 9 and Fig. 11 left
- color range of the degree of anisotropy for each of the above lines, Fig. 11 left
- ANSA XML result file that can be post-processed in META, Fig. 11 right
- draw elements per tensor values, Fig. 11 middle



```
<?xml version="1.0" encoding="UTF-8"?>
<AnsaXMLDocument>
    <!--*****************************************-->
    <!--File Name:C:/Users/fotopoup/Downloads/aaa/aaa-->
    <!--*****************************************-->
    <Dataset Name="Fiber orientation tensor on elements" ID="4010">
        <DataType> ELDT(Element data)</DataType>
        <DeptVar Name="Fiber orientation tensor on elements" Unit=""/>
        <NumberOfComponents> 6</NumberOfComponents>
        <NumberOfIndpVariables> 1</NumberOfIndpVariables>
        <IndpVar Name="Normalized thickness" Unit=""/>
        <Blocks>
            <NumberOfBlocks>5</NumberOfBlocks>
            <Block Index="1">
                <IndpVar Name="Normalized thickness" Value="-1.0" Unit=""/>
                <NumberOfDependentVariables> 5246</NumberOfDependentVariables>
                <Data>
                    <ElementData ID="4">
                        <DeptValues>      7.6903e-02        6.6758e-01        2.555
                    </ElementData>
                    <ElementData ID="5">
                        <DeptValues>      5.1088e-02        6.9370e-02        8.795
                    </ElementData>
                    <ElementData ID="6">
                        <DeptValues>      6.7886e-02        1.3731e-01        7.948
                    </ElementData>
                    <ElementData ID="7">
                        <DeptValues>      7.1057e-01        2.7429e-01        1.514
```

Fig.11: Visualization methods in ANSA for checking mapped fiber orientations.

In addition, after initialization with the solver ANSYS® LS-DYNA® post processes like Animator or META can be used for checking the mapped information on the component.

5 Material modeling of short fiber reinforced thermoplastics (SFRT)

The integration of process chain results in crash simulation imposes high demands on material model and mapping process. The material model plays a central role as it must be able to consider the local influence of process parameters onto mechanical material characteristics including viscoelastic, visco-plastic and strain rate dependent failure behavior. In case of short and long fiber reinforced polymers the material properties can additionally vary between highly anisotropic and isotropic states. Furthermore, material properties can vary significantly with in the crash relevant temperature range.

The phenomenological material model MF GenYld+CrachFEM has been developed to correctly consider different types of process chains which occur in a full vehicle crash simulation today [2, 7]. Within this study the crash simulation of injection molded parts using the user material model MF GenYld+CrachFEM in combination with explicit solver ANSYS® LS-DYNA® and a mapping by software ANSA will be shown (Fig. 12).

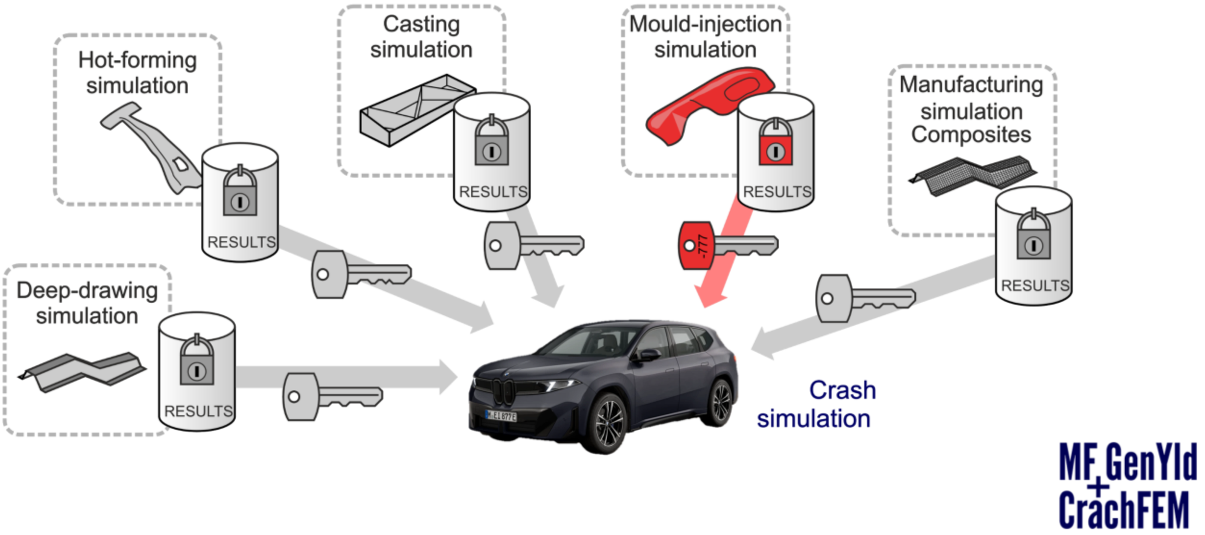


Fig.12: Integration of process chain results in crash simulation by using the user material model MF GenYld+CrachFEM in combination with ANSYS® LS-DYNA®.

The modular material model approach allows the application for a large variety of different materials. In the specific case of a short or long fiber reinforced polymer a subset of modules is used for the material representation (Fig. 12).

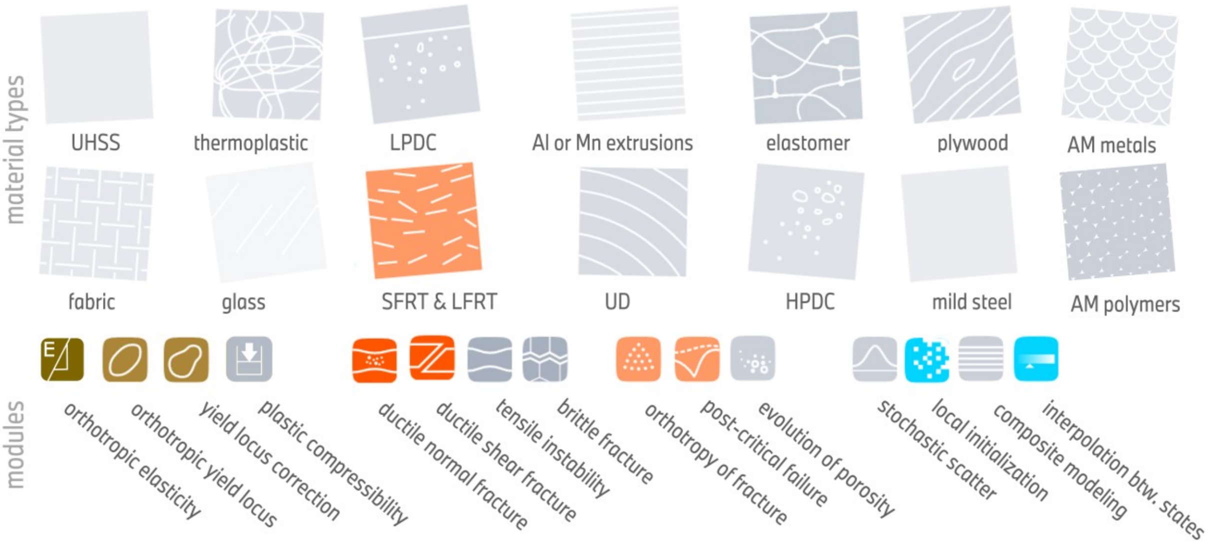


Fig.13: Material modeling of short or long fiber reinforced polymers using a subset of modules of MF GenYld+CrachFEM.

The material characterization is based on suitable testing campaign. Material and model parameters are identified in a systematic way at room temperature. This experimental design can be reduced appropriately for low and high temperatures in a range relevant for crash load cases [8].
The material model is used within the 3 different areas, characterization, visualization and simulation as displayed in Fig. 14. Based on a comprehensive material testing plan at different test speeds, and a specific temperature, material parameters are identified for a defined degree of fiber orientation.
During the process of material parameter determination, the modules of the material model are used for the fitting of material parameters. Visualization of the material properties in MF View, which uses the material model for the visualization, makes even complex material cards easily accessible and user friendly. The visualization in MF View not only allows to compare different material cards to one another but also facilitates plausibility checks of simulation results.

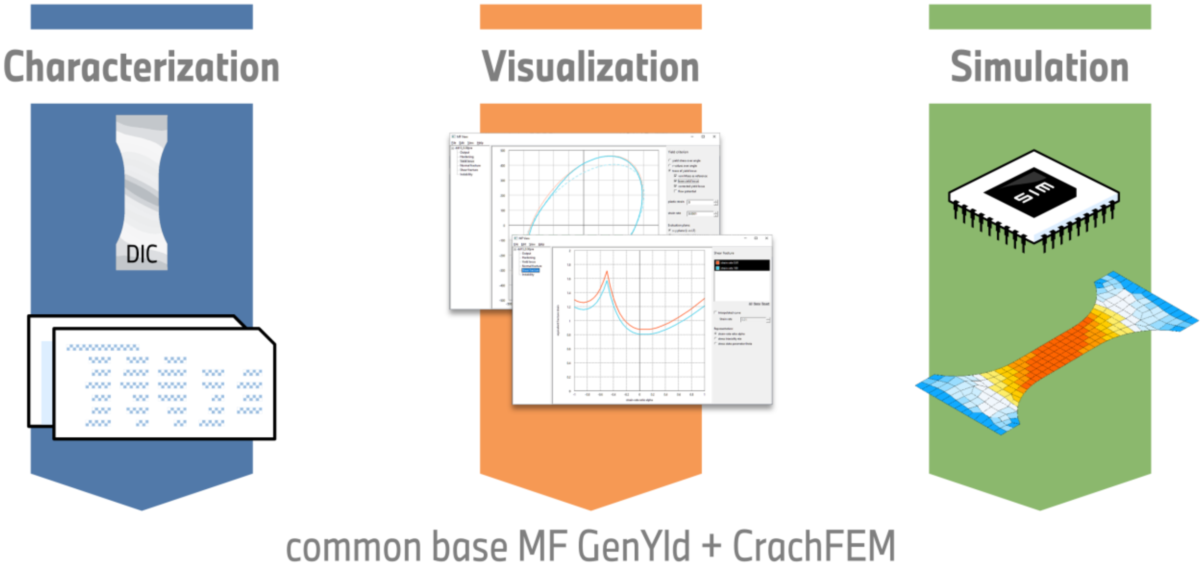


Fig.14: Material model MF GenYld+CrachFEM as a common base for material characterization, visualization and simulation.

Based on this material card the material model is hence capable of covering the visco-elasticity, visco-plasticity and fracture and their dependencies on strain-rate, fiber orientation and degree of anisotropy. The degree of anisotropy is a statistical measure for the fiber orientation state and allows to distinguish between high orientation and quasi random distribution fiber state. This is a crucial feature because locally the material properties within a part may vary greatly. [1]

Furthermore, the material model contains a stress-state dependent hardening model which allows to distinguish between different types of loadings (tension, compression, biaxial tension and shear loading). This comprehensive material card for a PP-LGF30 material for MF GenYld+CrachFEM can interpolate between isotropic and orthotropic material state based on the mapping information. Fig. 15 summarizes the visualization of visco-elasticity, yield locus and 3D fracture model for two different orientation states – isotropic and orthotropic.
Per default, isotropic material properties are assumed. This is a helpful feature when no mapping information is available e. g. in an early design stage or when no process simulation is available for a minor change in the design, e. g. the re-positioning of a rib or a flap. Figure 6 displays the corresponding visco-elasticity, yield locus and fracture orthotropy for an isotropic and orthotropic material state.

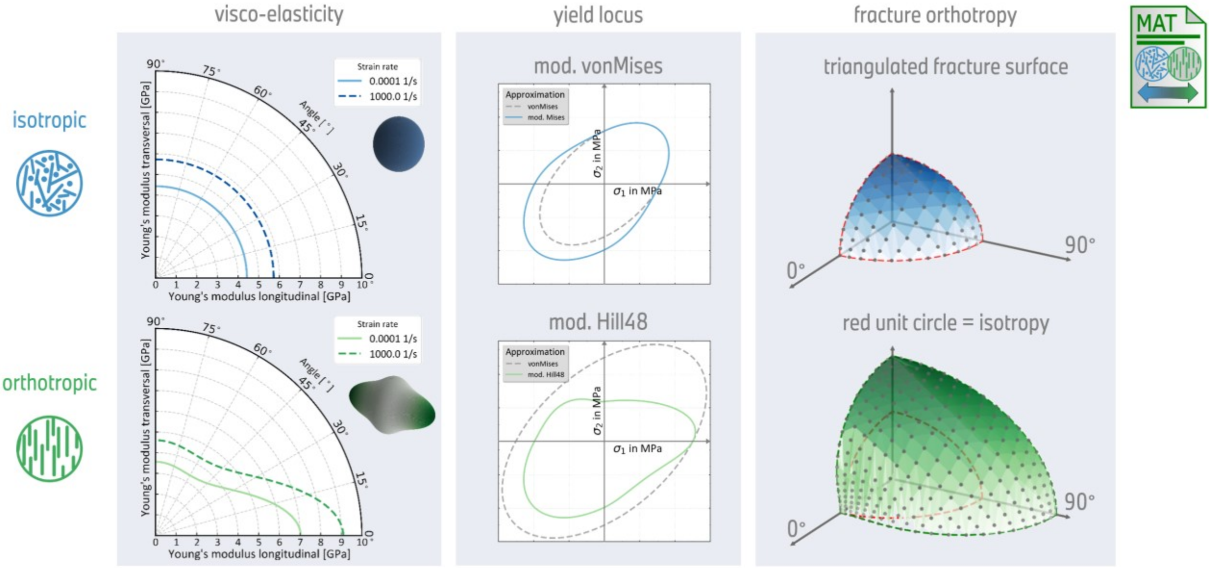


Fig.15: Schematic representation of two extreme material states for isotropic and orthotropic material states as bases for continuous interpolation in MF GenYld+CrachFEM.

Fig. 16 and Fig. 17 summarize the simulation results with the material card for multiple local strain rates and fiber orientation across uniaxial tensile test, puncture test, 3-point bending and compression test. The overall objective is to find a global optimum across all tests considered in the material characterization rather than a perfect fit for a single test as e. g. the uniaxial tension. This validation simulation on coupon level constitutes the last step of material characterization. To move further on component level the process history of the injection molding (IM) process containing is an essential prerequisite to obtain simulation results of similar quality.

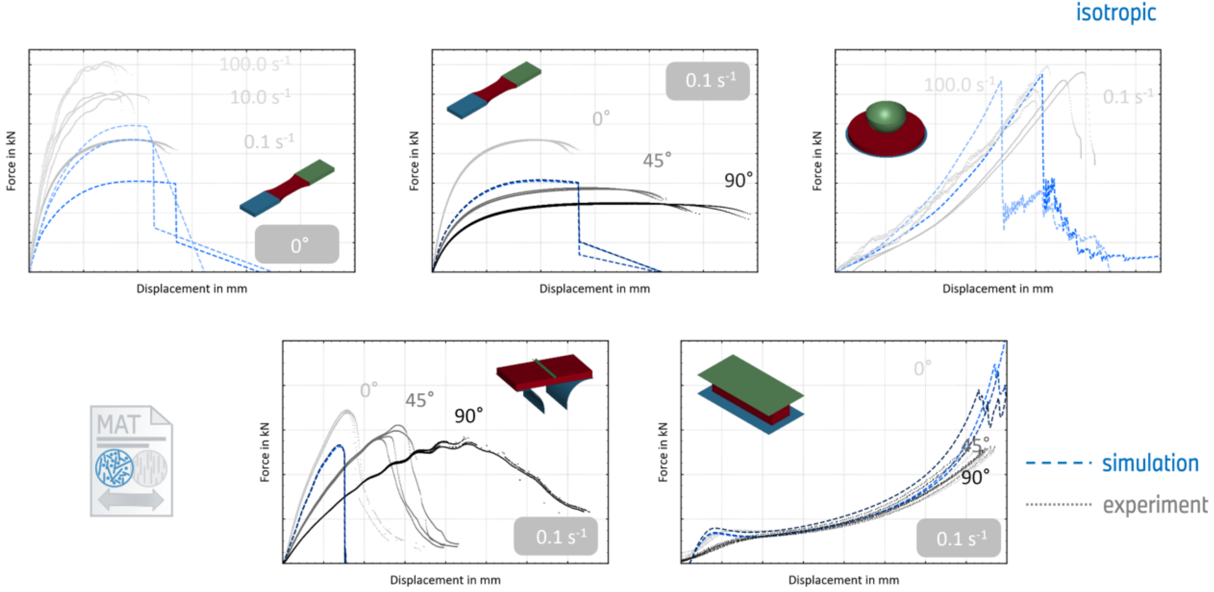


Fig.16: Comparison of force-displacement curves between experiment and simulation for uniaxial tension, puncture tests, three-point bending and compression tests – isotropic without mapping.

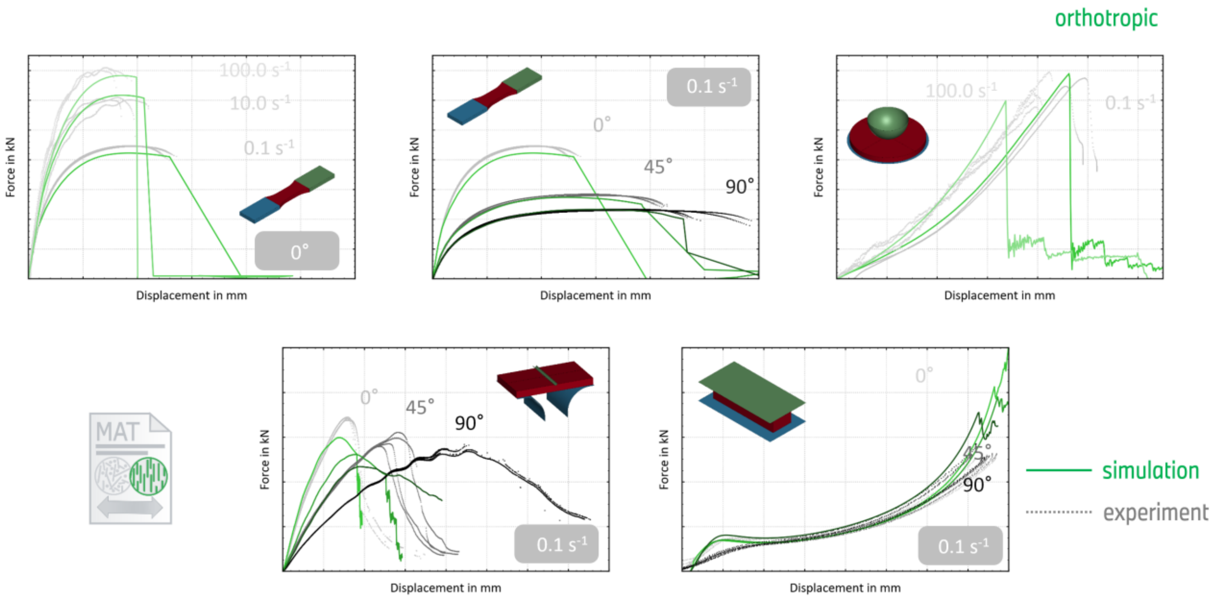


Fig.17: Comparison of force-displacement curves between experiment and simulation for uniaxial tension, puncture tests, three-point bending and compression tests – orthotropic with mapping.

The mapping file from ANSA is used to account for the history of the injection molding (IM) process simulated from an incremental third-party software for injection molding simulation. This file contains local information of the fiber orientation, degree of anisotropy and possible occurrences of weld lines as displayed in Fig. 18. The mapping is done individually for every integration point for e. g. shell elements covering the local microstructure of the fibers, such as e. g. the skin-core effect. For a correct modeling of weld lines material specific modification factors are calibrated experimentally and used locally to influence the material properties of the integration points in the vicinity of a weld line [3, 8]. Benefits and good simulation results of this powerful approach to include the markedness of the local microstructure of tiny fibers and to link them to macroscopic material parameters on big, geometrical complex components with industry requirements on computational time and discretization have already been published in [1] and [8].

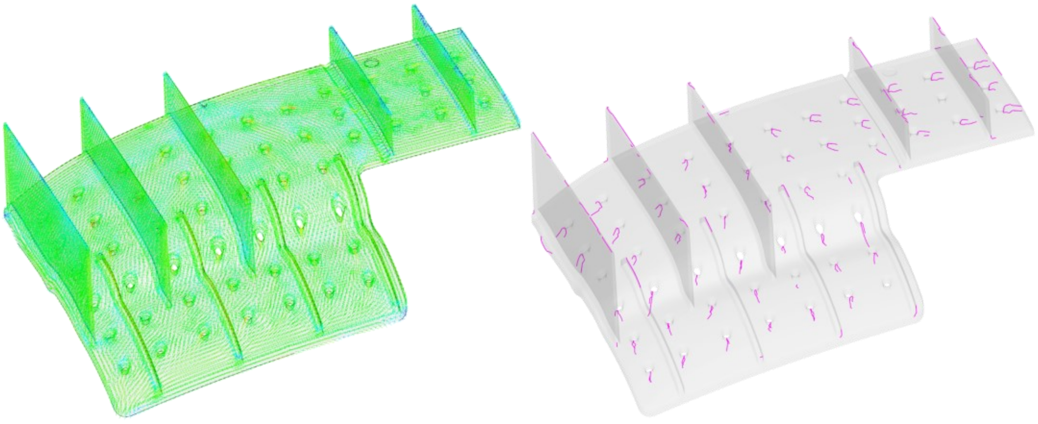


Fig.18: Mapped fiber orientation on the left and weld lines on the right on component level of rib absorber mapped with ANSA V25.1.

6 Results on part level

Using a simple component, which is used in the field of the headliner, the influencing factors in modeling are compared with the influences in the choice of the material model and the quality of the material information. To exclude factors that are not directly related to the component as far as possible, a component test was chosen. The following factors were investigated:

1. modeling of the clamping
2. influence of the geometry accuracy
3. choice of material model and corresponding material parameters

In the experiment the rib absorber as displayed on the right side in Fig. 1, Fig. 18 and right side of Fig. 19 was clamped on the left side and loaded with round bar of r = 20.0 mm at multiple loading velocities.

6.1 Modeling of the clamping

In the belief that the clamping has only a minor influence compared to the other factors, a simple modeling was started with a rigid body and a fixed node connection “Clamp old”. However, a comparison between simulation and tests very soon showed that if a better material model is used and the process information is taken into account, the modeling of the clamping must also be improved. Therefore, the clamping fixture used in the experiment was modeled, see Fig. 19 “Clamp rev”.

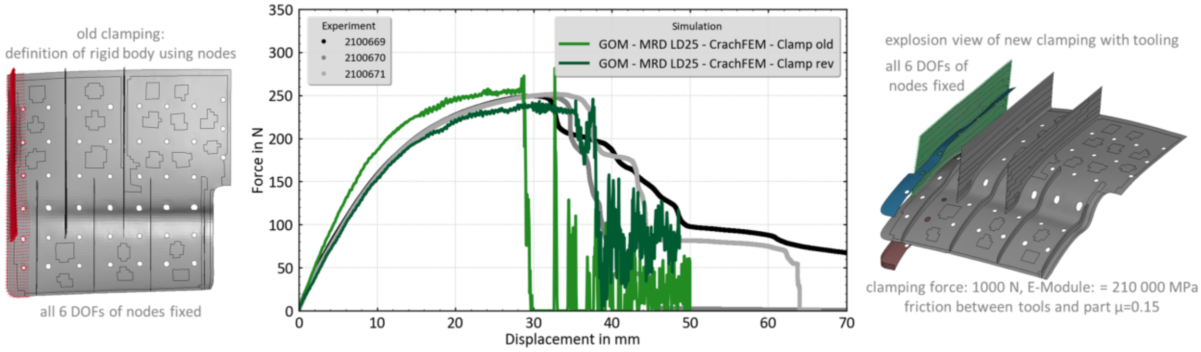


Fig.19: Modeling quality of clamping and its sensitivity.

A comparison of the measured force-displacement-curve during the test with the simulated force-displacement-curve using the real wall thickness and the material model MF GenYld+CrachFEM with the weld line and fiber orientation mapping shows the different bending stiffnesses and deformation paths at failure due to the modeling differences in the clamping.
The improved modeling of the revised clamping “Clamp rev” shows lower stiffness and later onset of failure, and the change of stiffness is comparable to the change of stiffness caused by the switch of the material model vonMises isotropic to CrachFEM isotropic, Fig. 21.

6.2 Accuracy of the geometry

Due to further deviations between the hardware tests and simulation, especially in the area of higher loads, the real geometry was measured using an ATOS 3D scanner – named as “GOM” – and the resulting point cloud was compared with the nominal computer-aided design geometry, see Fig. 20 right. This showed that the base wall thickness of the component in the mold parting surface deviated by more than 7%.
In the simulation, the different wall thicknesses were compared with the material model **`*MAT_PIECEWISE_LINEAR_PLASTICITY`**, i.e. the standard material model of ANSYS® LS-DYNA® using the improved clamping modeling.

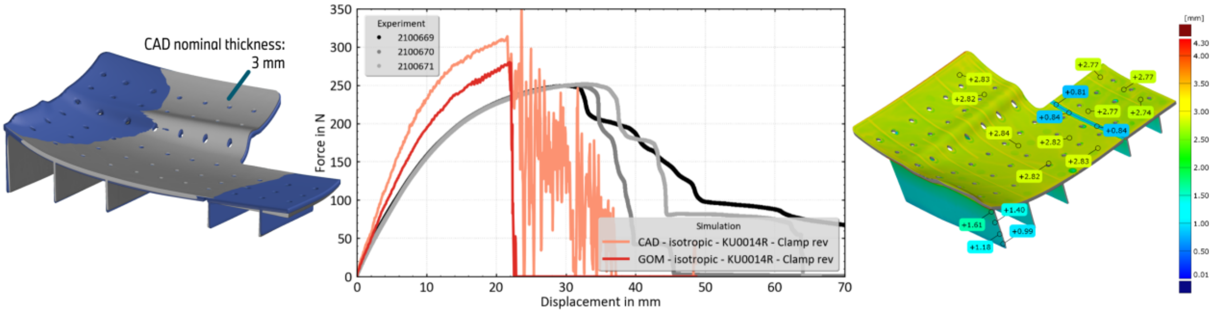


Fig.20: Accuracy of the geometry and its sensitivity.

The corrected wall thickness changes the stiffness comparable to the switch of the material model from vonMises isotropic to CrachFEM isotropic, see Fig. 21.

6.3 Material model and material card

The material models and parameterizations discussed in section 5 are compared with the help of the simulation model with improved clamping and corrected base wall thickness. This comparison shows that the consideration of the tension-compression asymmetry in the CrachFEM isotropic material model (Fig. 21, blue line) for the bending load present here already leads to a significant improvement of the material behavior. A further improvement - albeit only a slight one in this case - can be achieved by taking the process information into account with CrachFEM interpolating orthotropic material model (Fig. 21, green line).

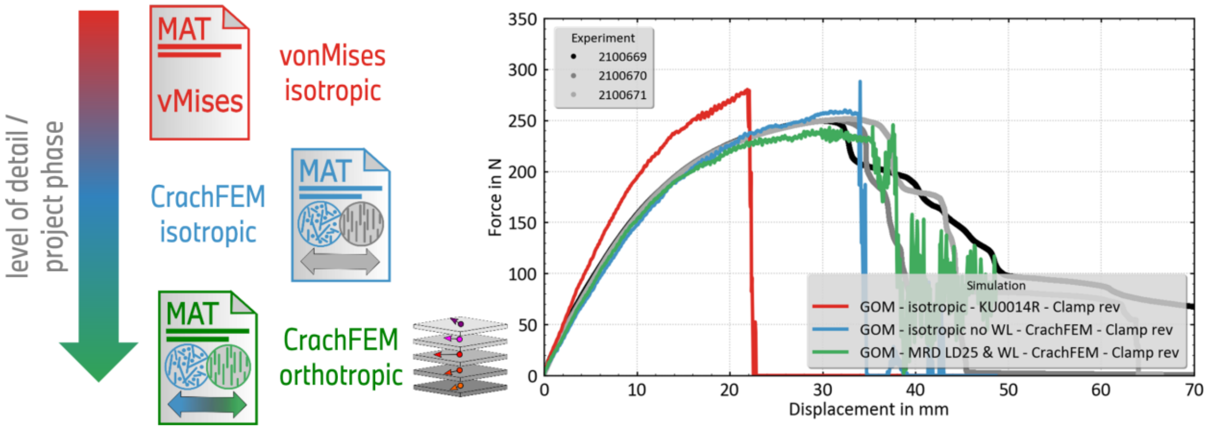


Fig.21: Changes due to the material model and parameterization quality.

The application of the different quality levels of the material cards to other components under various support and load conditions shows that the differences between CrachFEM isotropic and CrachFEM anisotropic can be significantly higher. As a result, it is necessary to consider from application to application which quality level makes sense - especially in connection with the remaining modeling quality.

7 Summary and Outlook

The mapping process with ANSA is successfully used for serial development at BMW for unreinforced as well as discontinuous fiber reinforced plastics. Important advantages are the modularity of MF GenYld+CrachFEM, the parametrization based on a physical framework and the usage in different stages of the development with one material card.
As shown at coupon and part level, the quality of process simulation, the mapping process itself, material model selection and material card implementation as well as the modeling of boundary conditions are equally important for good prognosis quality in crash simulation.
Visual checks and inspections are integrated at relevant points in the test module to ensure consistent quality. The meaningful visualization of process information, automated checks of individual modules, such as material cards and the results from the injection molding simulation, increase the user friendliness and have proven their worth.
For a successful roll-out and acceptance of the method, a precise description of the interfaces between the programs as well as reliable and stable support from our development partners are essential.
Due to the successful implementation for the inclusion of manufacturing process information for injection molded components, work is underway to extend this method to other industrial processes such as foaming and filling with bead foams, coatings, aluminum die casting as well as various sheet metal forming and sheet metal heat treatment processes.

8 Literature

References should be given in the last paragraph of your manuscript. Please use following scheme:

[1] Gross, M.: “Integrative crash simulation of short fiber reinforced thermoplastics (SFRT) – Investigation of phenomenologically based material modelling”, 2019, VDI PIAE
[2] Kaiser, Wolfgang. „Kunststoffchemie für Ingenieure: von der Synthese bis zur Anwendung.“ 6., Aktualisierte Auflage. München: Hanser, 2024.
[3] Schröder, Thomas. „Rheologie der Kunststoffe: Theorie und Praxis.“ 2., Aktualisierte und Erweiterte Auflage. München: Hanser, 2020.

[4] Tucker, Charles L. III. “Fundamentals of Fiber Orientation: Description, Measurement and Prediction.” Munich: Hanser Publishers, 2022.

[5] Schröder, Thomas. „Simulation in der Spritzgießtechnik“. Hanser eLibrary. München: Hanser, 2022.

[6] Wang, Maw-Ling, Rong-Yeu Chang, und Chia-Hsiang Hsu. “Molding Simulation: Theory and Practice.” 2nd edition. München: Hanser, 2022.

[7] Stommel, M.: "FEM zur Berechnung von Kunststoff- und Elastomerbauteilen", 2. Auflage, 2018

[8] Douven, L.: "Defining and validating MF GenYld+CrachFEM material model for an SFRP material", Proceedings of 6th MATFEM Conference, 2024