

On The Simulation Of Short Fiber Reinforced Engine Components

Tamás Schmidt

Engine Development Department of Audi Hungaria Motor Ltd.

Győr, Kárdán street 1. 9027 Hungary

Abstract: Lightweight technology is a very important part of today's engineering practice. The discussion of CO₂ emissions and fuel consumption gets more and more important. Hybrid vehicles could save fuel, but are also heavier. AUDI has made a sign with innovative new technologies in the hybrid models Q5, A6, A8, and has shown with the new A6 and A3 that a car could be lighter than its previous release. All these leading technologies are summarized in Audi Ultra. There are several ways to get a lighter construction. For example with new materials and technologies, or to perform more realistic simulations to use the capabilities of the material as much as possible. Another way is to optimize structures and manufacturing processes. This paper is attended to describe the advanced technology of simulating short fiber reinforced plastic engine parts. With the coupling of injection molding simulations and structural FEA it is possible to use the anisotropic material behavior resulted from fiber orientation. Identification of manufacturing process induced weak points becomes possible and a more realistic behavior of the part can be predicted. By the simulation of composite parts much more questions have to be considered. Problems of mesh density, material model description, tolerances, failure methods and indicators may arise earlier than in case of simulating metallic components.

Keywords: Automotive, Anisotropic Material Model, Composites, Converse, Digimat, Injection molding, Eigenfrequency Calculation, Failure, Fiber Reinforced Composite Material Model, Finite Element Simulation, Integrative Simulation, Micromechanics, Moldflow, Orientation Mapping

1. Introduction

There are many ways to reduce the fuel consumption of a passenger car. One is to optimize the engine with several techniques. Friction reduction of rotating and sliding components, a more effective combustion process, intelligent thermo management are some possibilities. Another way is lightweight technology like Audi's own concept called "ultra". This includes all the knowledge how to use innovative materials and intelligent design principles in resource-efficient production processes. In the future it is also desired to get the overall car producing process CO₂ neutral.

With the innovative lightweight methods Audi has turned back the weight spiral. The newest A3 and A6 models have – depending on the version – up to 80kg less weight as their previous releases.

Regarding the engine, the less loaded parts are usually made of plastic. Several components with short fiber reinforcement can be found in any modern engine, but their weight percentage is low.

2013 SIMULIA Community Conference

www.3ds.com/simulia

For example the new generation 1.8l R4 Otto engine has about 5% composite parts. Although it has more parts and is more complex, the weight was reduced about 3,5kg compared to the older release.

To use all the advantages of fiber reinforced composite parts, the simulation methods have to be able to describe their behavior under different loading conditions. Components have to be used at their limits to save as much weight as possible. This requirement meets the whole complexity of fiber reinforced plastics beginning from the manufacturing, up to dimensioning against high cycle fatigue loads under continuous material degradation due to influence of oil and cooling water.

This work will present a method how to use simulated fiber orientation data in structural analysis. Influences of mesh type and density, material model calibration will also be investigated. Practical experiences will show the benefits of the integrative simulation process.

2. Injection Molding Simulation Data

A realistic structural simulation on composite parts needs detailed information about the geometry, the loads and boundary conditions, and of course the material. Possible inaccuracy of loads and boundaries will not be investigated.

Most questions arise concerning about the material properties. Simulation of metallic components does not need too many assumptions for material definition. This is because steel and aluminum are almost homogeneous and isotropic. Results of stress and modal calculations are usually in good agreement with measured strain gauge and acceleration data.

Engine mounted lightweight components are usually reinforced with fibers. This means a two phase material of a polymer matrix and some kind of a fiber. This composite behaves strongly anisotropic. The mechanical and thermal constants of each component have to be known. With Micromechanical approaches the composite material properties can be defined.

For composite layers with continuous fibers the Rule of Mixture gives good results for the linear elastic material constants. Other micromechanical models have to be used in case of short fibers. These are according to the theory of Eshelby from 1957 which gives an analytical solution for a single inclusion in a homogeneous matrix.

For nonlinear elastic-(visco)plastic material behavior advanced analytical solutions, like the Mori-Tanaka homogenization model have to be solved. These approaches must be programmed in an Abaqus user subroutine which solves the equations during the analysis. Commercial mapping software have this subroutine included.

Micromechanical material models have to be coupled with fiber orientation data. The easiest way to obtain this information is to perform an injection molding simulation. The fiber orientation then has to be mapped onto the structural FEA mesh. Because of the different analysis type, the structural mesh is usually of a different element type or mesh size than the injection molding one. Figure 1 shows an example of the different mesh sizes. During mapping an orientation averaging has also been performed. The accuracy and reliability of injection molding simulation has an important effect on the structural simulation performed afterwards.

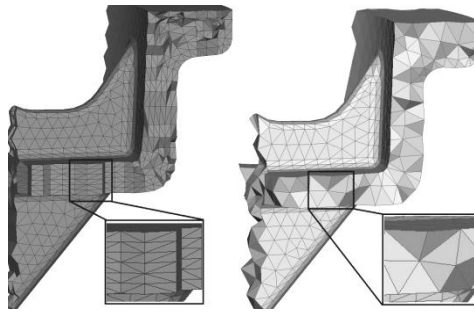


Figure 1. Different injection molding (left) and structural (right) meshes.

3. How to Use Fiber Orientation Data

3.1 Data exchange procedure

To represent the fibers via local element orientations and to get the resulting anisotropic stiffness, the injection molding results have to be transferred into the Abaqus model. Depending on the users technical resources and preferences several ways for data exchange are available. Figure 2 shows the process in general.

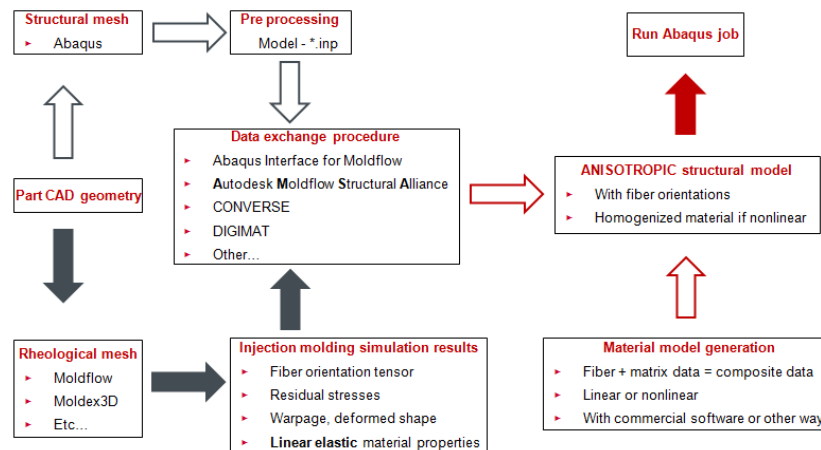


Figure 2. Overall data exchange procedure.

The following interfaces can be used for data exchange:

1. Abaqus Interface for Moldflow,
2. Autodesk Moldflow Structural Alliance,
3. Converse (from PART Engineering GmbH),
4. Digimat (from e-Xstream engineering),
5. Other commercial or self made algorithms.

Converse and Digimat are stand alone commercial software. Orientation mapping and anisotropic material property data generation is possible with them. Different injection molding and structural mesh types and sizes are also handled by these programs. Interfaces to Abaqus allow a quick and straightforward workflow.

Injection molding software writes the orientation tensor in a text format. Thus self made routines can also be written for transferring it to a structural mesh. Further details can be found in the work of Advani, S. G. and Tucker, C.L..

In this work the capabilities of Converse are used for data exchange. Figure 3 shows an example for a fiber orientation mapping on a tensile specimen. Represented is the degree of orientation in the first direction. Moldflow results are on the left hand side, the mapped orientations are on the right. In this case the Moldflow mesh has about 20.000 tetrahedral, the structural one only 860 hexahedral elements.

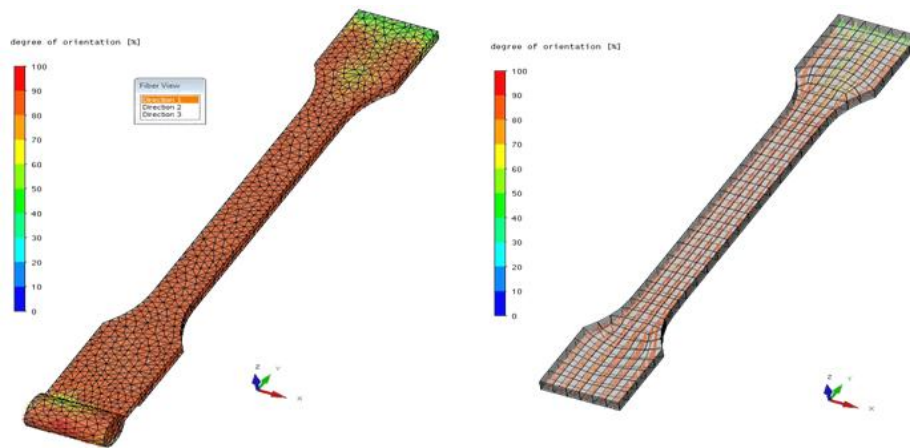


Figure 3. Orientation mapping on a specimen.

3.2 Composite material data evaluation

Composite material data can be obtained from the grain producer directly or from databases like CAMPUS. Data is commonly provided in fiber direction for a tensile test specimen after ISO 527-1/-2. Test results perpendicular to the main fiber direction are desired but not usually accessible.

Linear elastic composite data can be generated according to micromechanical methods by combining the basic mechanical properties of matrix and fiber. An example is made for a PA66 material with 35% mass fraction of glass fibers. The fiber diameter/length ratio is assumed to be 24. The datasheet Young modulus is 6 650 MPa. Table 1 shows the basic fiber and matrix properties. The evaluated elastic constants for a transversal isotropic case with different methods can be seen in Table 2. The calculated density of 1.142 g/cm^3 is independent from these methods.

Table 1. Fiber and matrix properties.

Property	Unit	Glass Fiber	Polymer Matrix
Young modulus	MPa	73 000	1 040
Poisson's ratio	-	0.22	0.34
Density	g/cm ³	2.55	1.147

Table 2. Calculated elastic constants.

Property	Unit	Rule of Mixture	Halpin-Tsai	Chow
E ₁₁	MPa	15 071	8 574	12 683
E ₂₂	MPa	1 287	1 874	3 712
ν_{12}	-	0.317	0.317	0.317
G ₁₂	MPa	481	557	428

Analytical solutions spread widely and give only the linear elastic constants. The fiber aspect ratio has a big influence on the results.

3.3 Calibration

Exact data for the fibers and the matrix is often not available. Tables of commonly used fibers can be used as a reference. Many grain producers also have the unfilled polymer version in their product portfolio. This is enough to identify initial material properties. Effects of behavior modifying additives cannot be represented in a simple material model.

Thus, a verification of the generated composite material model based on the datasheet curves is necessary. For this an injection molding simulation of the specimen with the desired material has to be performed.

First, the Young modulus of the composite has to be evaluated. This can be done for several temperatures. Afterwards the nonlinear viscoplastic part of the stress-strain curve has to be adjusted. Fitting the calculated curve on the test one needs many loops. Handmade try and error loops could be replaced with a time saving automated optimization using Isight.

Computation time increases significantly with the use of the user subroutine for nonlinear material definition. A comparison with a homogeneous isotropic material model is shown in Table 3.

Table 3. Comparison of computation time.

	Isotropic Homogeneous Nonlinear	Anisotropic Coarse Mesh Linear	Anisotropic Coarse Mesh Nonlinear	Anisotropic Moldflow Mesh Nonlinear
Element Type	C3D8	C3D8	C3D8	C3D10
Number of Elements	860	860	860	15 599
Number of Variables	4 218	4 218	4 218	76 695
Number of Increments	11	11	60	213
Factor for Slow Down	-	1	24	919

3.4 Failure prediction

Beginning failure in form of cracks is not allowed in automotive engine area. Parts are undergoing cyclic loading during their lifetime so any propagating crack could lead to an uncontrolled failure. This means that static failure indicators derived from the tensile test are sufficient for dimensioning. The main goal is to identify critical points of the structure in an early phase of the development process.

In case of a nonlinear material model, the user subroutine separates outputs for the homogenized macroscopic material and the matrix. Following values can be evaluated at the breaking point of the stress-strain curve:

1. Von Mises stress for composite,
2. Equivalent plastic strain for composite,
3. Von Mises stress and strain in the matrix.

In case of an anisotropic but linear material model, commonly known failure criteria can be used to identify critical areas. These formulas are divided into two groups. One gives information about the failure mode. Another provide just an overall value if failure occurs or not. The commonly used formulas are for example:

1. Maximal stress or strain criterion,
2. Tsai-Hill,
3. Tsai-Wu.

These methods were developed for continuous fiber reinforced composites. Because of this they are just applicable in case of a plane stress state, like on a part's surface.

These failure criteria are available in Abaqus but only for plane stress continuum shell and membrane elements. A Python script is needed to calculate these formulas for three dimensional elements. The results can be visualized in Abaqus/Viewer. This technique uses stress data from a completed Abaqus job. It has no effect on the simulation progress, like element deletion for example.

4. Practical Experiences

4.1 Modeling technique

The attempt for weight reduction results in more complex geometrical shapes with thin walls. Several functions can be integrated in composite parts. Also different designing rules have to be considered as by metallic parts.

From the meshing side this means that lots of tetrahedral elements are needed. The general rule, that walls, ribs should have at least 3 or 4 elements across their cross section, leads to very high element numbers.

Engine mounted parts are rarely modeled with shell elements. 3D tetrahedral elements are used to reproduce such a difficult geometry. Advanced hardware capabilities allow the usage of these element type instead of a hardly to generate hexahedral mesh.

On the other hand, engine parts cannot be investigated standing alone. The influence of surrounding components has to be taken into account. Pre-loading states, like bolt assemblies are part of the standard simulation procedure. The user always has to care about the overall FE model size. This in many cases leads to coarse tetrahedral meshes with often just 2 elements across the wall-thickness. Just like on the right side in Figure 1.

Figure 4 shows a thin layer of wedge elements on the parts outer surface. These are used to apply the failure criteria described in chapter 3.4. The inside is meshed with tetrahedrons. In this way we get slightly more elements.

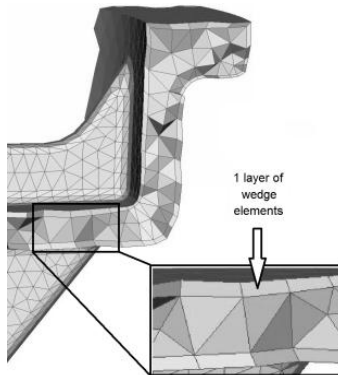


Figure 4. Layered mesh with wedge elements.

The mapped fiber orientations are defined per element in Abaqus. Each element has one orientation defined on all its integration points. This can be seen on Figure 5. This way each element can be investigated on its own. Figure 6 shows an example of this method on the tensile specimen.

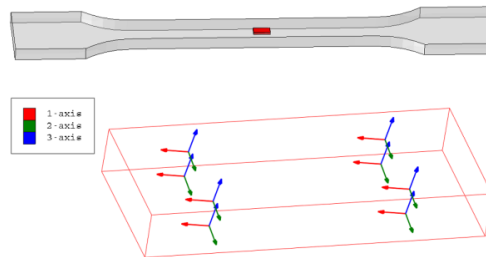


Figure 5. Fiber orientation of an element.

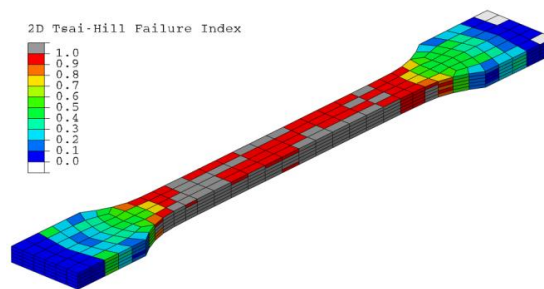


Figure 6. Tsai-Hill failure criterion results.

4.2 Modal analysis

Injection molding and structural meshes cannot be the same. The rheological one is commonly finer. Therefore some orientation information gets lost during the mapping process. The difference can be determined with a free vibration analysis without any boundary conditions. Defining the error made if the fiber orientations and the anisotropic stiffness are not taken into account is also desired. In the homogeneous isotropic case the datasheet Young modulus is reduced. It is assumed that the original injection molding mesh gives the most realistic result. Therefore the calculated eigenfrequencies will be compared to it. Figure 7 shows the first eigenmode of the investigated parts.

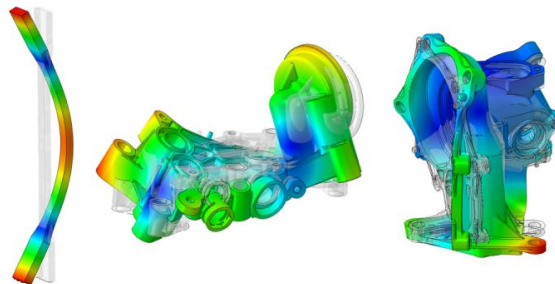


Figure 7. Investigated parts.

4.2.1 Tensile specimen

This example shows that the calculated eigenfrequencies for nearly same mesh densities but different element types are almost the same. The hexahedral mesh with only 860 elements seems to be too coarse. Results are shown in Table 4.

The datasheet Young modulus has to be reduced by 36% from 6 649 MPa to 4 255 MPa to capture the same first eigenfrequency with a homogeneous isotropic material model. In this case the second eigenmode is captured within a tolerable error, but from the 3rd one the error becomes higher.

Table 4. Results for tensile specimen.

	Unit	Moldflow Tetrahedral	Coarse Hexahedral	Tetrahedral	Wedge	Coarse Hexahedral
Material Model		anisotropic	anisotropic	anisotropic	anisotropic	homogeneous isotropic
Number of Elements		15 599	860	21 876	12 384	860
1 st mode	Hz	330	357	330	330	331
2 nd mode	Hz	682	683	682	681	670
3 rd mode	Hz	791	792	791	791	954
4 th mode	Hz	984	1044	984	984	1046
5 th mode	Hz	1952	2059	1952	1952	1895

4.2.2 Part A

Different mesh densities are investigated in this example. Figure 8 shows the element numbers for each variant. The injection molding mesh in Moldflow has 3,3 million tetrahedral elements. The desired coarse structural mesh contains 472 thousand tetrahedral elements.

Results represented in Figure 9 show that the mesh density has practically no influence on the calculated eigenfrequencies. This means that computation time can be saved with a relative coarse structural mesh.

The Young modulus has to be reduced by 55% in case of an isotropic simulation to capture the first eigenfrequency of the Moldflow mesh. Considering that raising the first eigenfrequency with 10-20 Hz is a great challenge in engineering practice, the difference about 51 Hz by the 2nd eigenmode between the reduced isotropic and the anisotropic calculation is large.

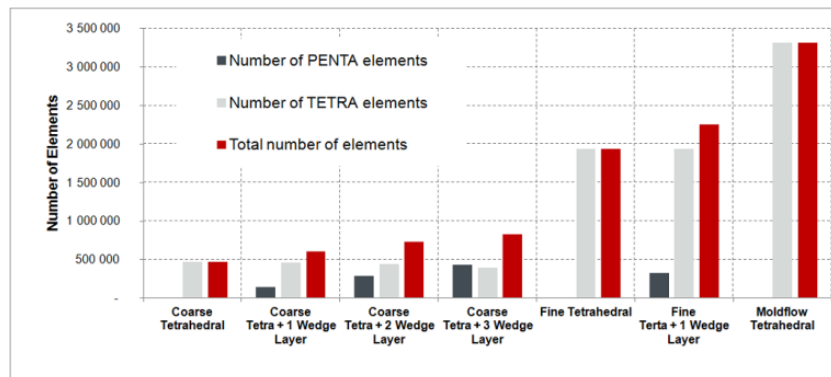


Figure 8. Element numbers of the investigated models.

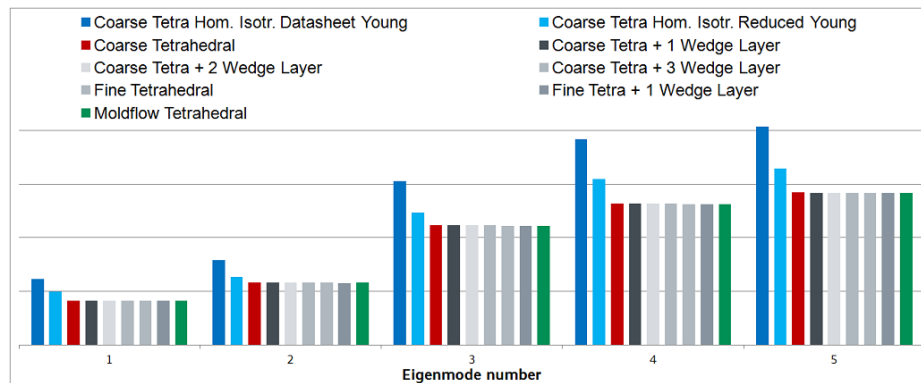


Figure 9. Comparison of the first 5 eigenfrequencies.

4.2.3 Part B

This part has a structural mesh with nearly 1.2 million elements. One layer of wedges was used and the interior contains tetrahedrons. The original Moldflow mesh has 2.9 million tetrahedral elements.

Table 5 shows interesting results. The used anisotropic structural model is far away from the injection molding values. This difference could come from the material calibration or from lost orientation data. With a 61% reduced datasheet Young modulus the first 5 eigenfrequencies are very close to the Moldflow ones. In this case the homogeneous isotropic calculation would be good enough.

Table 5. Results for Part B.

	Unit	Moldflow Tetrahedral	Tetrahedral	Datasheet Young Modulus	Reduced Young Modulus
Material Model		anisotropic	anisotropic	homogeneous isotropic	homogeneous isotropic
1 st mode	Hz	659	736	1063	664
2 nd mode	Hz	1007	1109	1588	992
3 rd mode	Hz	1306	1450	2090	1305
4 th mode	Hz	1456	1594	2295	1433
5 th mode	Hz	1701	1904	2741	1711

4.3 Stress analysis

Stress results must be interpreted carefully. If an orientation is defined for an element, Abaqus writes out the results by default in this local coordinate system. In case of an anisotropic linear material model the stress components are the following:

- S11 – stress in fiber direction.
- S22 and S33 – stresses perpendicular to fiber direction.

If the analysis was performed with a nonlinear material model using the user subroutine, additional stress data is available. These are described in chapter 3.4.

4.3.1 Tensile specimen

The tensile test is used for material calibration and for evaluation of static failure indicators. Figure 10 shows the investigated mesh types. Simulations were performed with a displacement based load of 4.5mm. The failure indicators were compared at the end of the analysis step. The stress-strain curves in Figure 11 and the values in Table 6 show that there is no significant influence of mesh size and type. Practically the hexahedral mesh with just 860 elements is enough for evaluating static failure indicators. This model has a clear performance advantage in the calibration of the nonlinear material model as described in chapter 3.3.

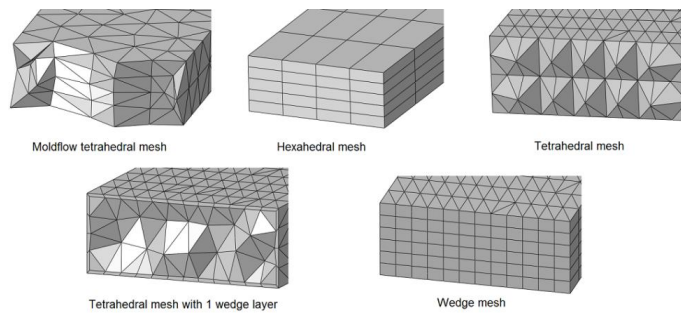


Figure 10. Investigated mesh types.

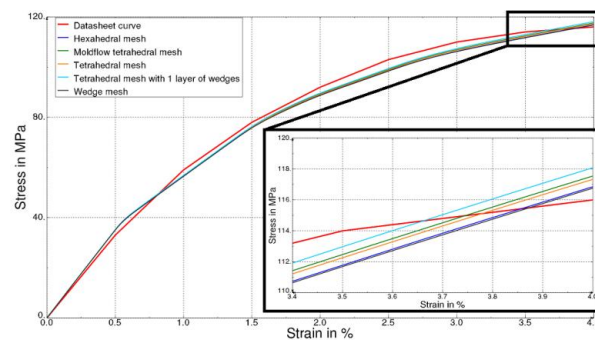


Figure 11. Stress-strain curves.

Table 6. Failure indicators for different meshes.

	Unit	Moldflow Tetrahedral	Hexahedral	Tetrahedral	Tetrahedral with 1 Layer of Wedge	Wedge
Number of Elements		15 599	860	21 876	24 066	12 384
Element Type		C3D10	C3D8	C3D10	C3D10, C3D15	C3D15
Von Mises Stress for Composite	MPa	150,9	137,0	144,1	147,7	148,7
Equivalent Plastic Strain for Composite	%	7,3	7,2	7,4	7,2	7,7
Von Mises Stress for Matrix	MPa	41,5	41,2	41,7	41,3	42,2
Factor for Slow Down	-	38	1	41	70	32

4.3.2 Part A

This example shows the difference between the tetrahedral and the layered mesh. The extreme high mishandling load is applied over a large arm. Goal is to identify the structure's weakest points. Figure 12 shows this model and the direction of the applied load.

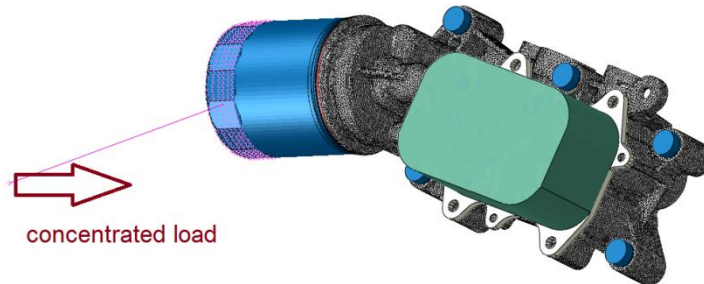


Figure 12. Assembly and loading conditions.

Table 7 summarizes the stress results in the most critical rib area. In case of a homogeneous model the Von Mises, for the anisotropic one the stress in fiber direction is evaluated. The results are represented in percentage of the anisotropic linear solution which is the commonly used method.

The tetrahedral mesh is coarse. In some areas only one element through thickness direction can be found. The overall element number of the model is about 500 thousand.

The layered mesh has the advantage of more integration points through the wall thickness. This is the reason for the higher stresses of this version. The table also shows that a homogeneous isotropic model under predicts the loading of the structure.

The difference between the two mesh types gets smaller in case of a nonlinear anisotropic material model. This analysis has the most realistic results but with the most computation effort.

Table 7. Stress results for different calculation methods.

Material model		Stress type	Unit	Tetrahedral	Tetrahedral with 1 Layer of Wedge
Isotropic Homogeneous Datasheet Young Modulus	linear	Von Mises	%	81	89
Isotropic Homogeneous Reduced Young Modulus	linear	Von Mises	%	73	79
Anisotropic	linear	S11	%	100	114
Anisotropic	nonlinear	S11	%	73	75

The 2D Tsai-Hill failure index is calculated with a Python script for each element separately. Results can be represented with a quilt type contour plot in Abaqus/Viewer. In this way each face

has a value extrapolated from the nearest integration points of the element. There is no averaging between elements. Figure 13 shows the failure indexes for the critical area. The values show the results at the element centroid. In an XFEM calculation the damage initiation parameter is also investigated at this point. The wireframe elements show the position of the nodes and the integration points marked with small black lines. The layer mesh follows the surface while tetrahedrons can extend deep into the part.

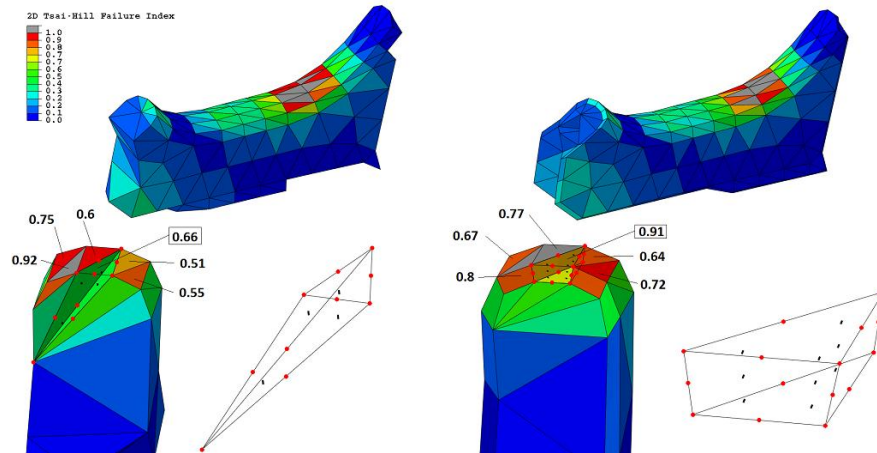


Figure 13. Results for tetrahedral and layered mesh.

5. Conclusions

Considering the fiber orientations of a composite part is necessary to obtain realistic results. Therefore homogeneous isotropic material models fail if stiffness prediction or a stress calculation has to be done. One method for transferring injection molding result into a structural FE analysis was presented. A complex material calibration is also part of this procedure. Failure indicators can be evaluated with a good accuracy on a relatively coarse hexahedral mesh in a very short time.

Orientation data is lost if the structural mesh has significantly less elements than the rheological one. The overall model size is in contrast with the mesh density and quality criteria. Availability of hardware resources and time is also a restriction.

Fiber orientations are essential for eigenfrequency calculations. There is no general rule how much the Young modulus of a composite in a homogeneous isotropic simulation has to be reduced to get the same results as with respect to the anisotropic stiffness. The value is geometry, process and polymer type dependent.

Interpretation of stress results with anisotropic material models depends on the used method. Nonlinear approach with the user subroutine increases computation time significantly. Therefore a calculation with linear elastic material properties should be performed first. Critical areas can be identified with or without calculating failure criteria. A layered mesh with wedge elements is able to capture the stress state near the outer surface of the part. Thus the Tsai-Hill failure criteria with

an element by element approach can be used. Another advantage is that more elements and integration points are defined through the wall thickness. The layered mesh is an acceptable choice in engineering practice for investigating short fiber reinforced composite parts.

6. References

1. Abaqus Analysis User's Manual, Versions 6.11-3 and 6.12-2, Dassault Systèmes Simulia Corp., Providence, RI.
2. Advani, S. G., Tucker, C.L. "The Use of Tensors to Describe and Predict Fiber Orientation in Short Fiber Composites", Journal of Rheology, 1987
3. Audi official website, www.audi.com
4. CAMPUS website, www.campusplastics.com
5. e-Xstream engineering website, www.e-xstream.com
6. FeResPost user's manual, www.ferespost.be
7. Mori, T., Tanaka, K. "Average Stress in the Matrix and Average Elastic Energy of Materials with Misfitting Inclusions". Acta metall, 1973
8. PART Engineering GmbH website, www.part-engineering.de
9. A. Radtke, „Steifigkeitsberechnung von diskontinuierlich faserverstärkten Thermoplasten auf der Basis von Faserorientierungs- und Faserlängenverteilungen“, Fraunhofer-Institut für Chemische Technologie, 2008
10. M. Stommel, M. Stojek, W. Korte, „FEM zur Berechnung von Kunststoff- und Elastomerbauteilen“, Carl Hanser Verlag, 2011