AUTODESK UNIVERSITY 2014

As-Manufactured Structural Simulation of Fiber-Filled Plastics

Doug Kenik – Autodesk

SM5135-P

In this class we will discuss the new capability to map fiber orientations and material properties from Simulation Moldflow software to structural finite element analysis (FEA). We will also cover how to use Simulation Composite Analysis software to simulate nonlinear material behavior of the fiber-filled material.

Learning Objectives

At the end of this class, you will be able to:

- Discover the new interface between Simulation Composite Analysis software and Simulation Moldflow software
- Understand the nonlinear material behaviors supported by Simulation Composite Analysis software
- Understand how to use Simulation Composite Analysis software for fiber-filled materials
- Learn how to confidently interpret results from Simulation Composite Analysis software and fiber-filled materials

About the Speaker

Doug Kenik was born and raised in Wyoming and attended college at the University of Wyoming where he pursued a Master's Degree in Mechanical Engineering studying micromechanics of composite materials. Directly after obtaining his Master's degree, he was employed by Firehole Technologies as a software developer, researcher, and applications engineer. Doug spent 5 years at Firehole, the last 2 of which were directed at account management and product development. Firehole was eventually acquired by Autodesk, and now Doug is a Product Manager for the composites software which was acquired from Firehole.

Introduction

Structural performance of short fiber reinforced plastics can be very difficult to predict. Specifically, fiber reinforced plastics exhibit nonlinear material behavior due a variety of factors, including (but not limited to):

- Plasticity
- Viscoelasticity
- Temperature
- Humidity

The inherent material nonlinearity present in fiber reinforced plastics is compounded with the fact that they may exhibit orthotropic material behaviors, depending on how they're manufactured. Fiber may align themselves with flow directions which can cause severe orthotropic behavior, they may be randomly oriented causing isotropic behavior, or they may exhibit any directional dependence between the two, depending on how the fibers are oriented.

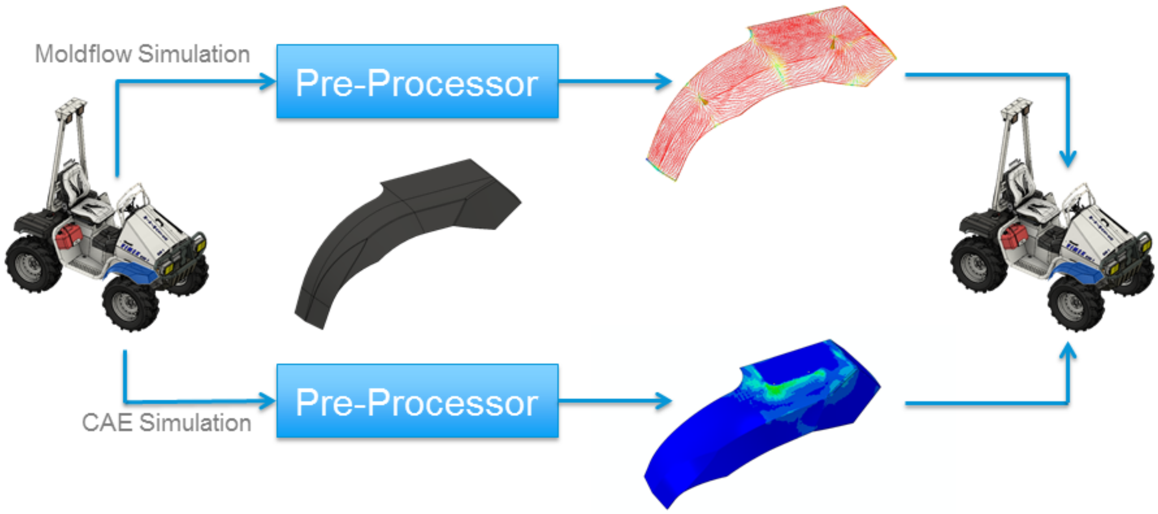
Last year at Autodesk University we debuted a new Labs offering called Project Sundance, which was aimed at simulating elastic-plastic behavior of fiber reinforced plastics during structural analyses under thermo-mechanical loading. Project Sundance was aimed at mapping fiber orientations and material properties from an Autodesk Moldlfow manufacturing simulation to an Abaqus structural simulation. Users could then predict nonlinear material behavior of their structural part within Abaqus by using the as-manufactured properties.

This year, we are proud to introduce the commercialization of Project Sundance within Autodesk Simulation Composite Analysis (ASCA). We have broken Sundance into two portions:

1. The front end, which maps fiber orientations, material data, and residual stresses/strains from a Moldflow mesh to a structural mesh is now called the Advanced Material Exchange (AME)
2. The back end, which applies material orientations, nonlinear material data, and (optionally) residual stresses and strains now works as a User-Define Subroutine (UMAT) in conjunction with the structural solver.

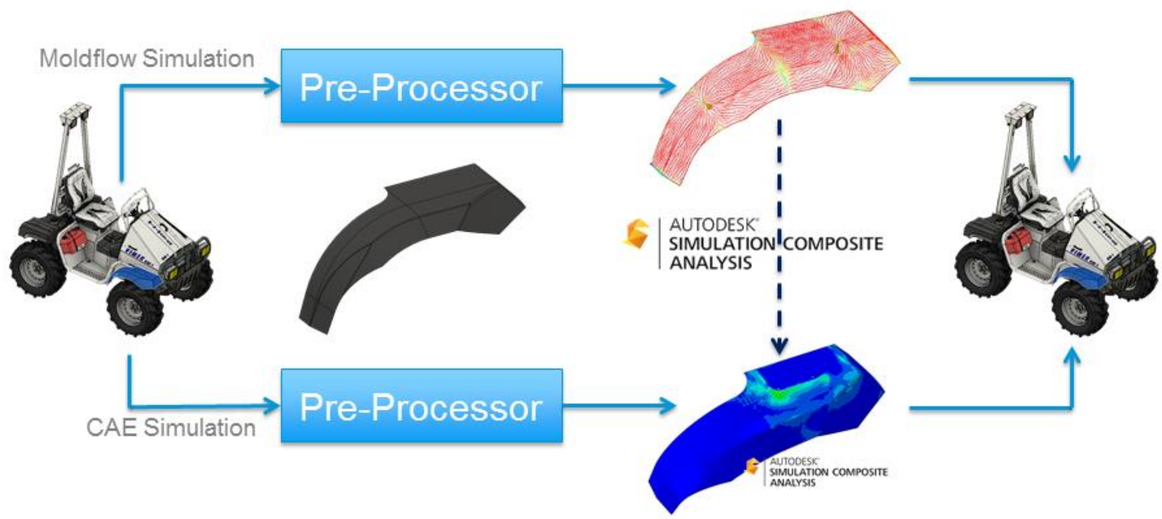
Transferring Data from Moldflow to a Structural Model

The workflows for structural simulation and Moldflow simulation can be very different, as shown in the figure below. Often times, the two simulation paths diverge. Modifications are made to the models to satisfy the needs of the different requirements. Once complete, the models converge and are updated to reflect the design changes made by engineering.



Often, the structural simulation neglects the ‘as-manufactured’ condition of the part, in that the material is assumed to be isotropic, and all residual stresses/strains (which cause warpage) are neglected.

AME is the new tool for mapping fiber orientations, material properties, and residual stresses and strains (e.g. warpage) from a Moldflow simulation to a structural mesh. Now engineers can collaborate to ensure structural simulations take into account and quantify he most recent manufacturing processes.



The most common problems between the structural and manufacturing simulation models can be summarized below:

1. Manufacturing simulations are typically complete on a part-by-part basis, and structural simulations are completed as-assembled
2. Manufacturing simulations and structural simulations do not share the same global orientations or positions
3. The two simulations use very different meshes. This includes both element types and mesh densities.

AME has been working on solving these problems to help in transferring data foe 'as-manufactured' simulations.

Assemblies

The idea of mapping as-manufactured data for multiple components which share the same structural model is well known. Manufacturing simulations are typically completed on a component by component basis, while the structural simulation requires taking the interaction of the global assembly into consideration.

Using the data on the solver side of the simulation is not (typically) a problem. All the solver really needs is a material description and orientation. The harder problem is setting up a user interface which can help users map the data from the Moldflow simulation to the structural simulation easily and seamlessly. Something that Autodesk has not solved yet, but we are working on diligently.

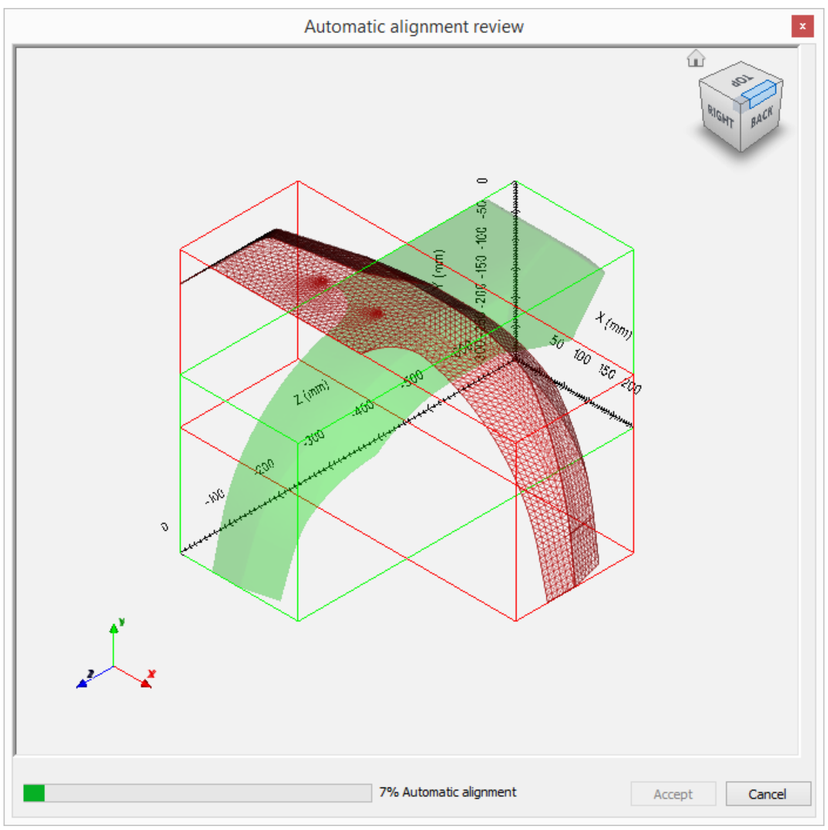
Aligning Models

Often, manufacturing simulations and structural simulations do not share the same global position. Moldflow simulations are typically oriented in 'Tool Space' which helps to identify positioning in relation to the tools used for manufacturing. The structural simulation, however, is oriented in the 'assembly-space' which coincides with the global positions and directions which can identify the overall structure. A common example of 'assembly-space' is the global position and orientation used for automobile simulations. Here, all components are positioned according to an orientation which defines the entire automobile structure.

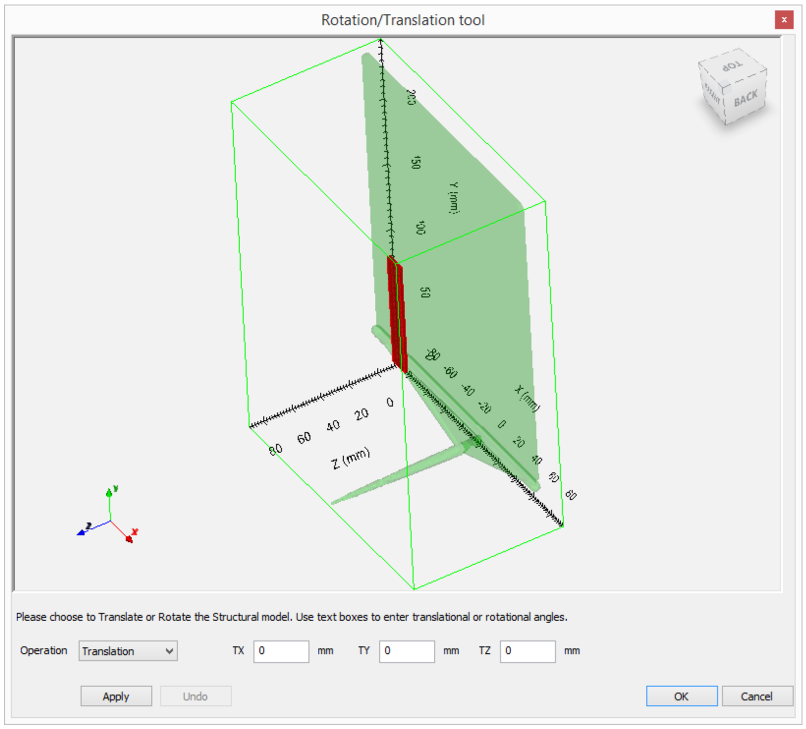
AME offers two methods for aligning models which do not share the same position:

1. Automatic Alignment
2. Interactive alignment

Automatic alignment is exactly what it sounds like! Automatically align the models without any user interaction. As long as the two components have the same general physical description, automatic alignment can usually align the models with ease (see below).



The other option to align models is interactive alignment. Interactive alignment is best used when one of the geometries is much different than the other. Take, for example, a simple tensile specimen cut from an injection molded plaque. The two geometries are totally different! But, the interactive alignment tool allows users to place the tensile bar within the Moldflow geometry and map the orientations by using a series of translations and rotations.



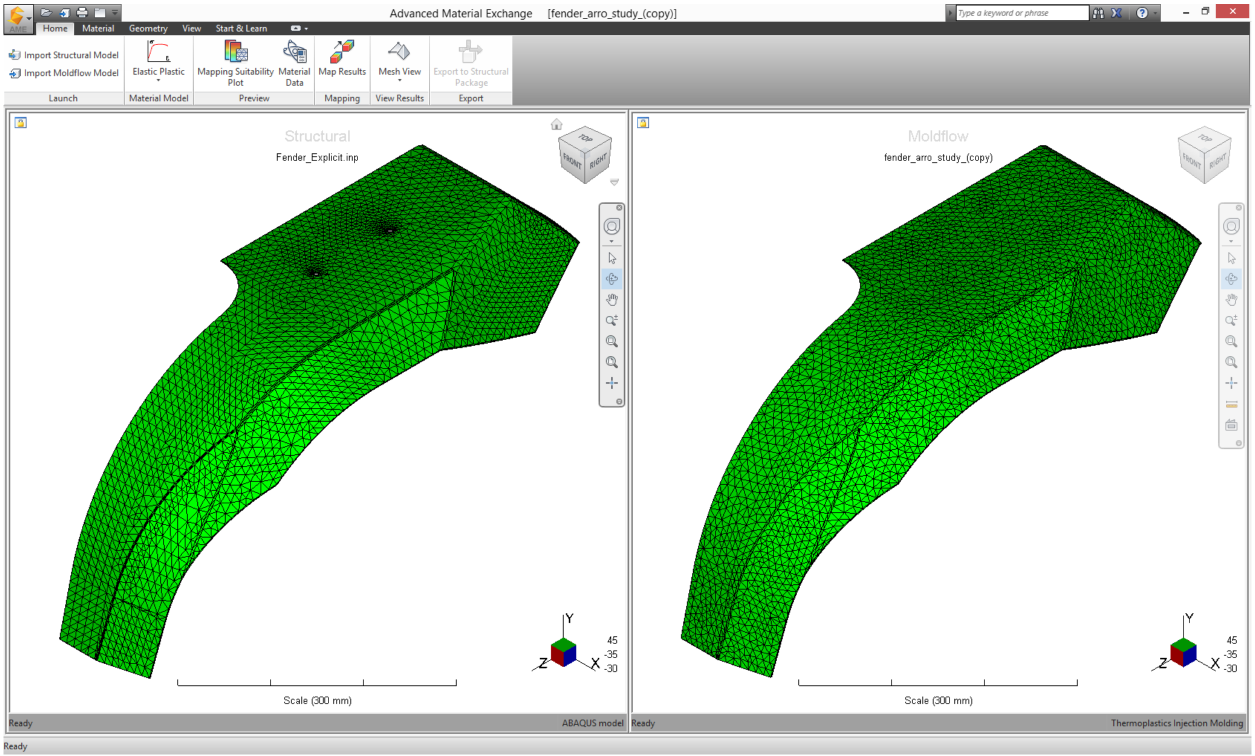
There is no restriction on the geometries being identical. AME will map the data from Moldflow to the structural mesh for whatever geometry shares the same position once aligned.

Mesh Compatibility

The final known problem for mapping ‘as-manufactured’ data between manufacturing and structural simulations is mesh compatibility. Typically, the two simulations do not share the same element type or nodal positions (mesh density). Often, the structural simulation uses a much coarser mesh than the Moldflow simulation and often uses higher order elements.

AME attempts to alleviate any mesh restrictions by allowing users to map data for a variety of element types with different mesh densities. Data from a 3D Moldflow simulation can be mapped to a variety of Abaqus 3D elements, including:

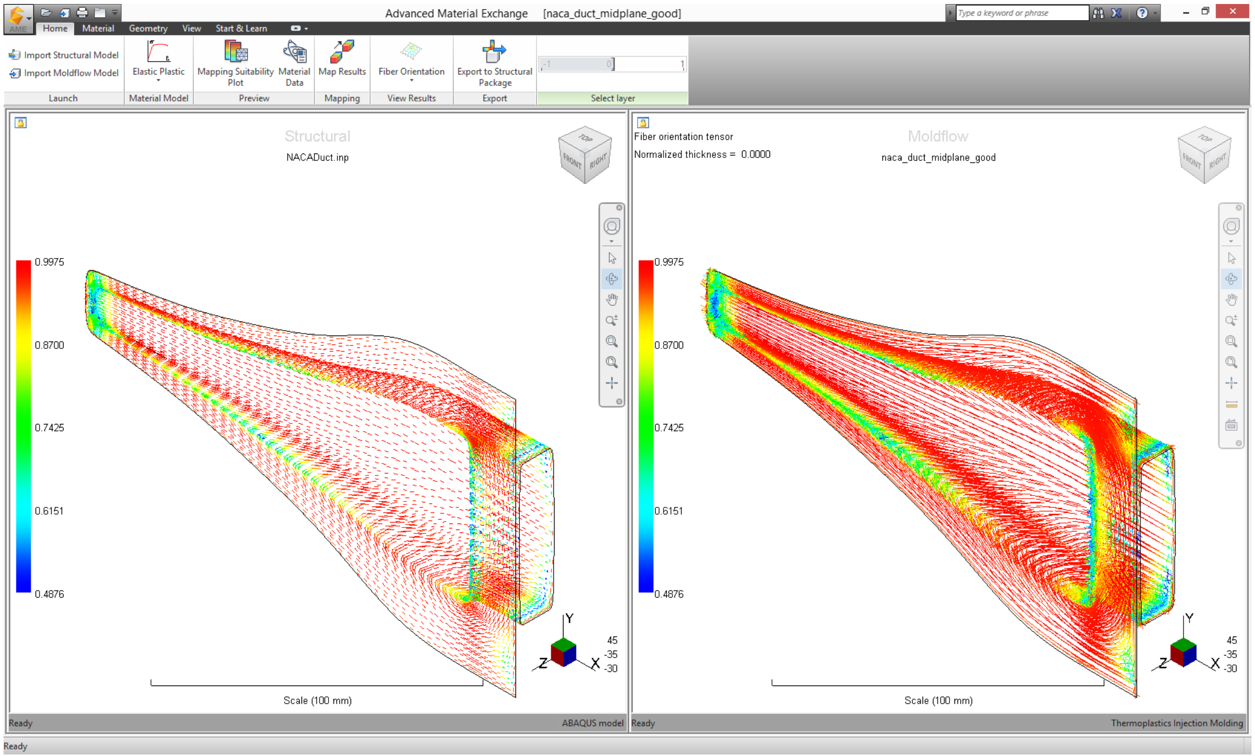
- 4 node tetrahedral (C3D4)
- 10 node tetrahedral (C3D10)
- 8 node brick - full integration (C3D8)
- 8 node brick - reduced integration (C3D8R)



Similarly, AME can map data for a mid-plane Moldflow simulation to an Abaqus shell mesh. The supported Abaqus shell elements are as follows:

- 3 node shell (S3)
- 4 node shell (S4)
- 4 node shell – reduced integration (S4R)

AME only supports Abaqus elements with the default of 5 section points through the thickness, and it will allow the user to interactively look at the fiber orientations through the thickness of each part side-by-side (top, middle, and bottom).



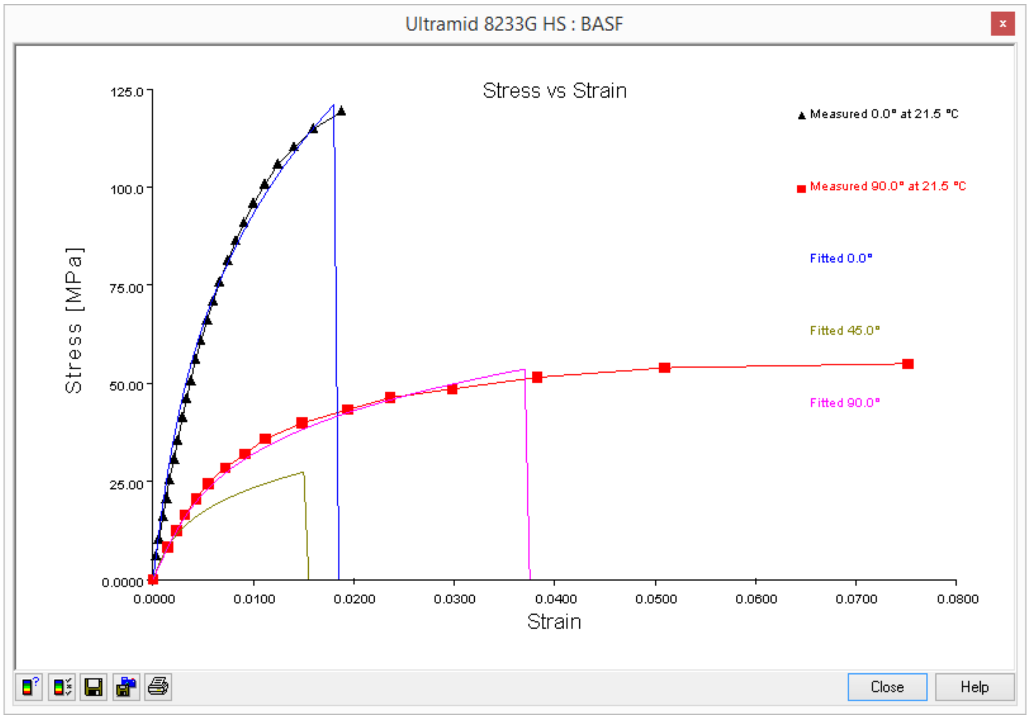
Material Nonlinearity

Fiber filled plastics exhibit nonlinear material behavior which can be a function of many different variables:

- Strain
- Time
- Temperature
- Humidity

For most mechanical simulations, structural simulations are interested in simulating the structural behavior of a part at a specified strain rate and humidity. The component might undergo both thermal and mechanical loading. These simulations are typically aimed at understanding if the material remains in the linear elastic regime during loading, or if plasticity and/or matrix rupture occurs during the loading process. ASCA is currently focused on simulating the elastic-plastic response (to rupture) of a fiber-filled material subjected to thermo-mechanical loading.

AME will transfer all material data from Moldflow to the structural mesh. In addition, it relies on user input to define the elastic-plastic behavior of the material to rupture by asking the user to input stress-strain curves to failure in both the flow and cross-flow directions. AME uses this data to reverse engineer the mechanical properties of the matrix constituent for use in the nonlinear simulation. For more information on the theory and reverse engineering process, please refer to the handout which accompanies this document.



On the solver (FEA) side, ASCA will use the data interpolated by the AME in the analysis through a user-defined material subroutine (UMAT). In simplest terms, UMATS are a material description that a user can program. The FEA program will ask the UMAT for a material stiffness matrix and stress tensor while supplying a strain tensor. This occurs for every integration point and at every equilibrium iteration in an analysis.

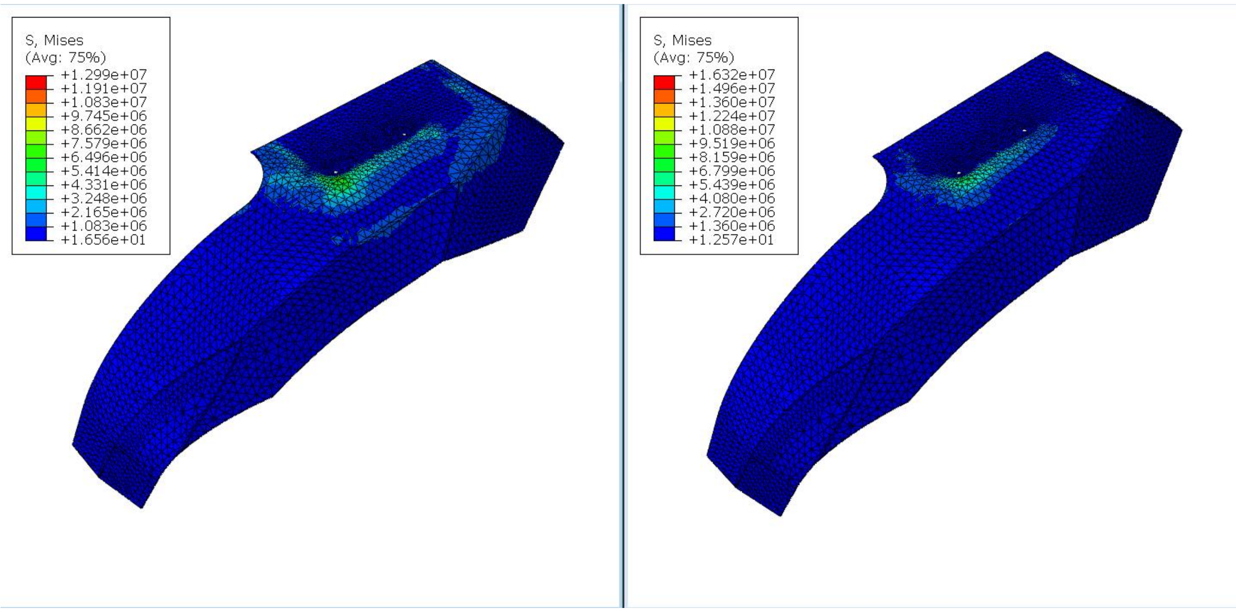
The benefits of using a UMAT to define the orthotropic nonlinear material behavior are three-fold:

1. There is no need to write out orientations for every element, which can bloat input files and significantly slow down pre-processing times.
2. There is no need to define an initial stress or strain state to include residual stresses or strains. The UMAT can define these quantities on the fly at the beginning of the analysis. The solver will automatically accommodate and deform the shape.
3. Material nonlinearity and failure is directly controlled using the expertise and algorithms of ASCA, which is tailored for robustness, efficiency, and accuracy.

Material Orientations

AME can define the material orientations for every integration point in the structural analysis. It passes this information to ASCA for use in the structural analysis by writing the data to an accompanying file (the .sif file).

ASCA can parse the file and acquire the material orientation tensor for every integration point at run time with minimal overhead in the structural analysis. So, for every integration point, ASCA will rotate all data to the material coordinate system, work with the data in this coordinate system, and rotate it back so the FEA program understands the consequences of the strain increment.



Comparison of von-Mises stress for a linear material model using as-manufactured material orientations (left) and an isotropic assumption (right),

Residual Stresses and Strains

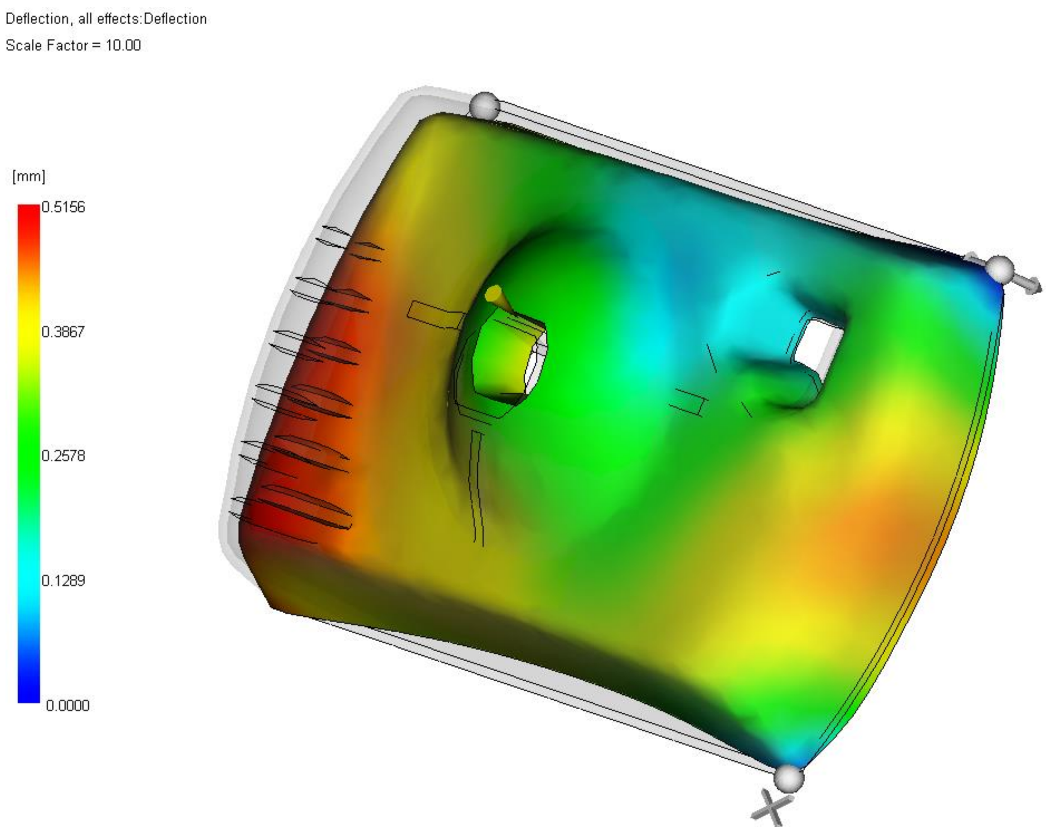
One of the most common requests from users is the ability to include residual stresses & strains from Moldflow into a structural simulation. Residual stresses and strains define the warpage of the part and can be used in a structural analysis to recreate the deformed part in Moldflow.

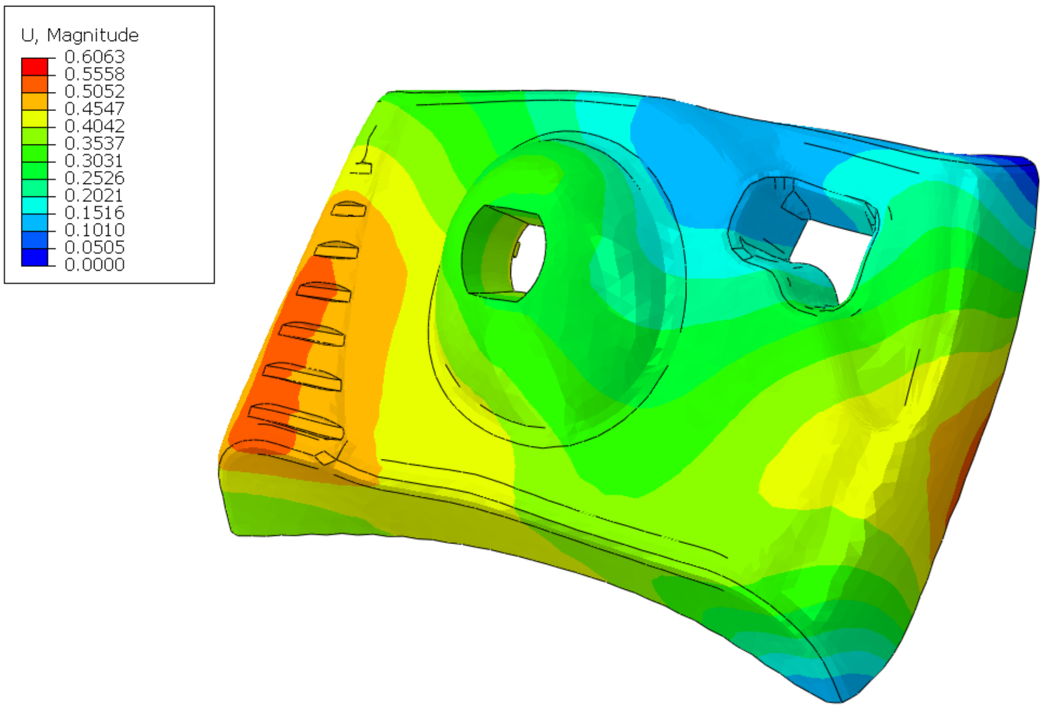
Like fiber orientations, the residual stresses and strains must be interpolated from one mesh to another such that they can be accurately defined at the integration points. The next step is applying them to the analysis. Residual stresses (or strains) are traditionally applied at the beginning of an analysis on an element by element basis through a pre-defined field. This is a fairly cumbersome process, and unfortunately is not the same (or supported) by all FEA codes.

Instead of using pre-defined fields, Autodesk Simulation Composite Analysis uses the interpolated values from Advanced Material Exchange. To understand the basics of how this can be accomplished we must understand a bit about a nonlinear FEA solver.

A nonlinear FEA solver's purpose is to achieve equilibrium (minimize potential energy) using a balance of residual forces at nodes. This is achieved using stress, strain, material stiffness, and external loadings (applied forces, displacements, body loads, etc.). Any time there is an imbalance in equilibrium, the solver will update the strain field (strains are a derivative of displacement) to try and match the previous known external loads, stiffness matrix, and stress field.

We can 'trick' the solver into applying the warped shape by directly applying the residual stresses and creating the imbalance in strain and stress (with a known stiffness). The only option for the solver is to equilibrate the system by forcing the model into the deformed shape. So, we can apply a step prior to any mechanical loading where the structure is simply supported similar to that of the Moldflow warpage prediction. If residual strains are included in the simulation ASCA will create the imbalance automatically and force the FEA solver to create the warped shape, with the residual stresses and strains intact (see below).





Material Nonlinearity and Failure

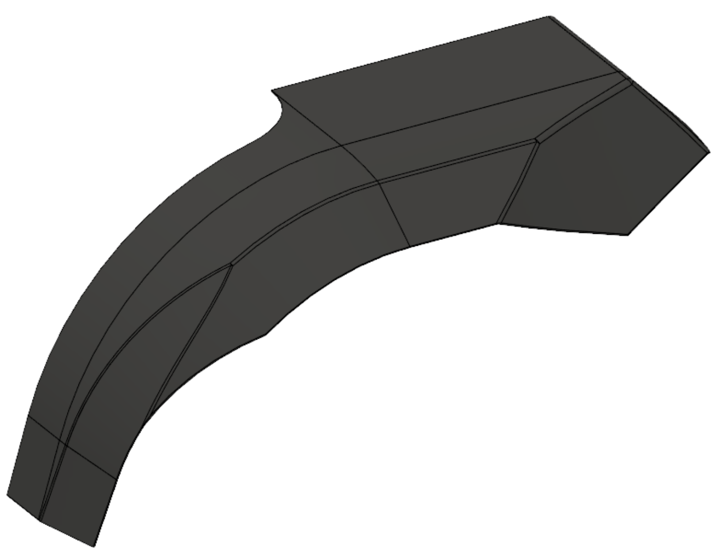
For a detailed description of the theory and supported material behavior of fiber reinforced plastics, please refer to the accompanying handout.

Use Case – As Manufactured Simulation of a Fiber-Filled Fender

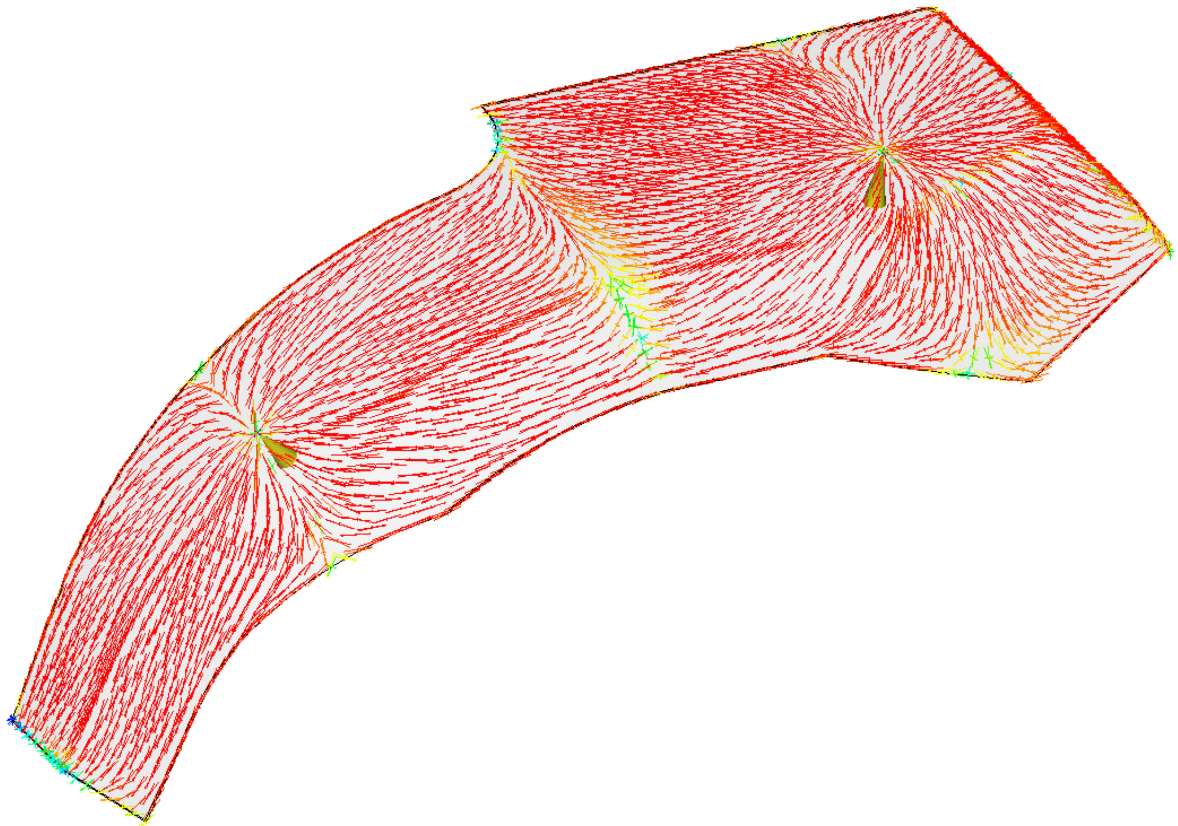
This case study is aimed at comparing a structural simulation of an injection molded fender for an 'as-manufactured' simulation to a linear isotropic simulation to demonstrate the importance of considering fiber orientation in structural simulations.

The fender belongs to a larger body of an automobile, shown below (the fender is highlighted in blue).

The CAD geometry of the fender is shown below. The CAD model was used for both the Moldflow simulation and the structural simulation in Abaqus.

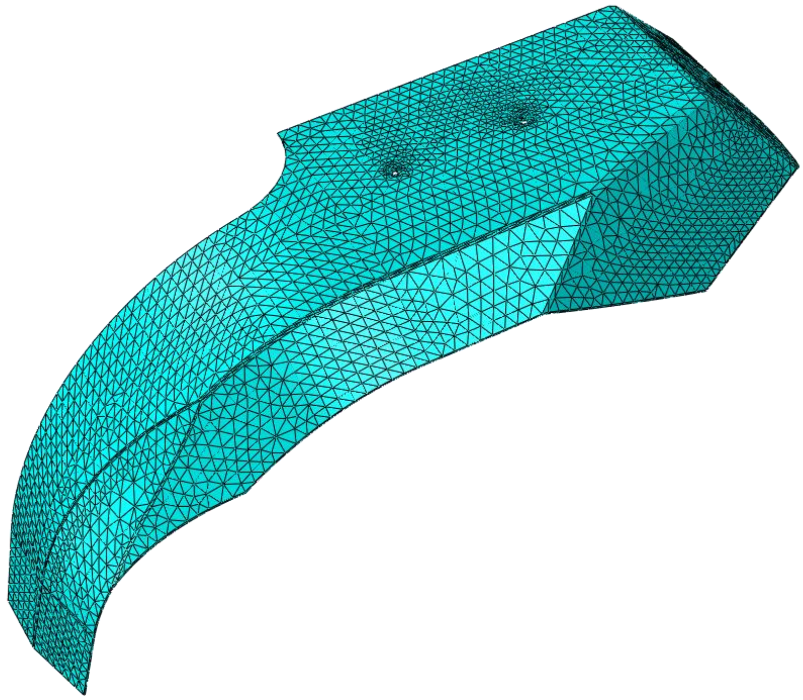


The Moldflow simulation used a 33% fiber-filled material from BASF (Ultramid 8233G HS) which was tested at the Autodesk Lab in Ithaca.



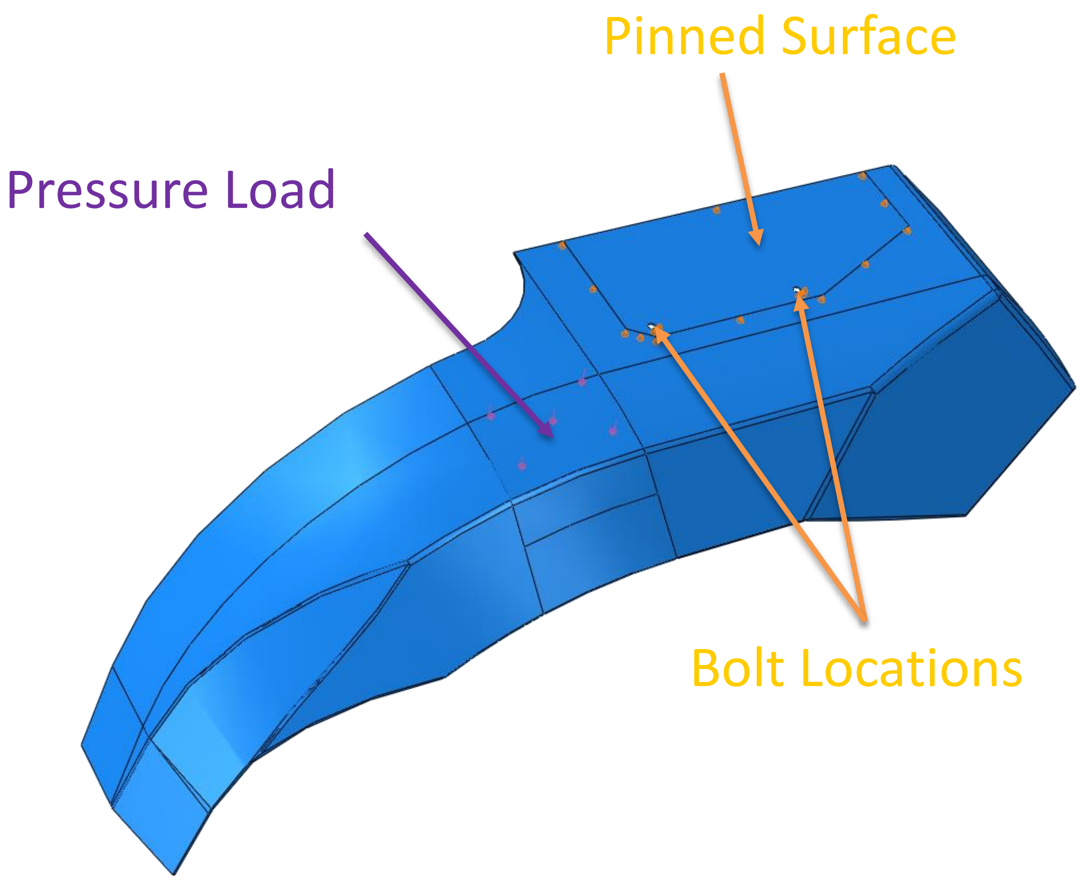
Injection point locations were chosen arbitrarily underneath the geometry to supply weld-lines and high degrees of fiber alignment, as shown above.

The Abaqus model was setup in Abaus CAE. The mesh is shown below:

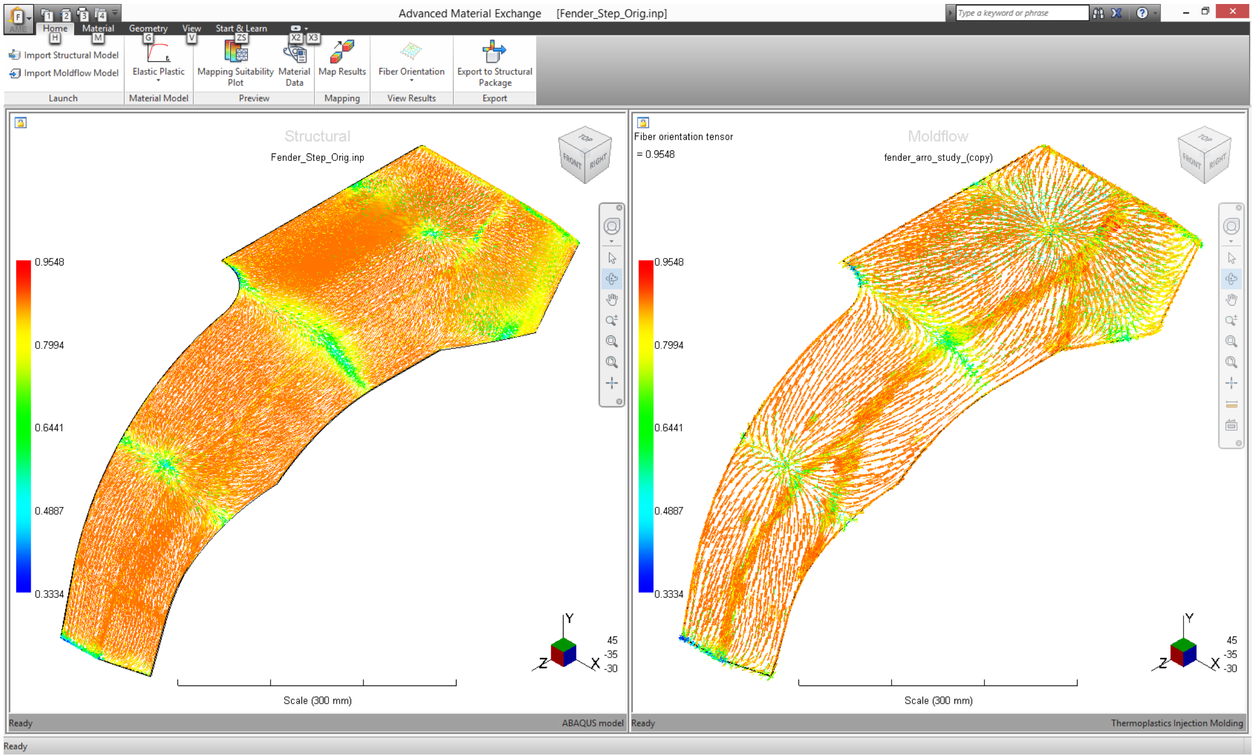


The loads and boundary conditions were setup as follows (pictured below):

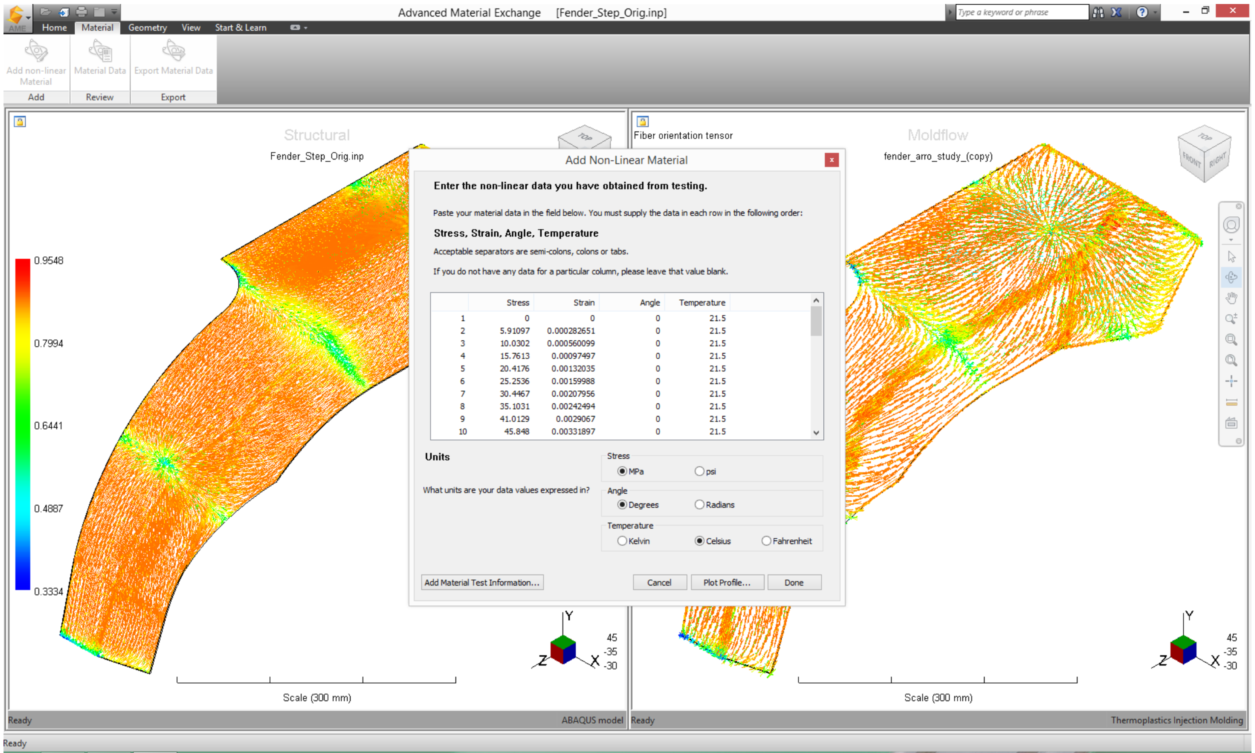
- Two bolt locations were fixed in translation (a gross oversimplification)
- An entire surface was pinned in the through-thickness direction mimicking the mounting plate for the automobile
- Pressure load of 2,000 Pa was applied to a small section representing a location which might be stepped on



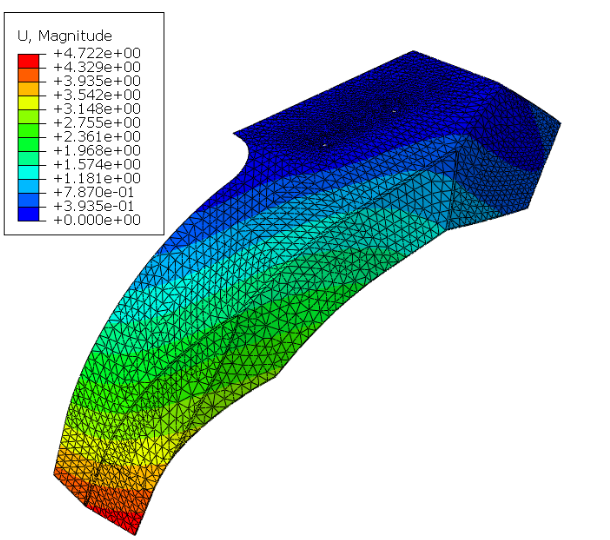
The Abaqus input file was exported from Abaqus CAE and imported to AME (Home > Import Structural Model) along with the Moldflow results (Home > Import Moldflow Model). AME automatically aligned the two parts, and we mapped the fiber orientations from the Moldflow model to the Abaqus model with one click (Home > Map Results).

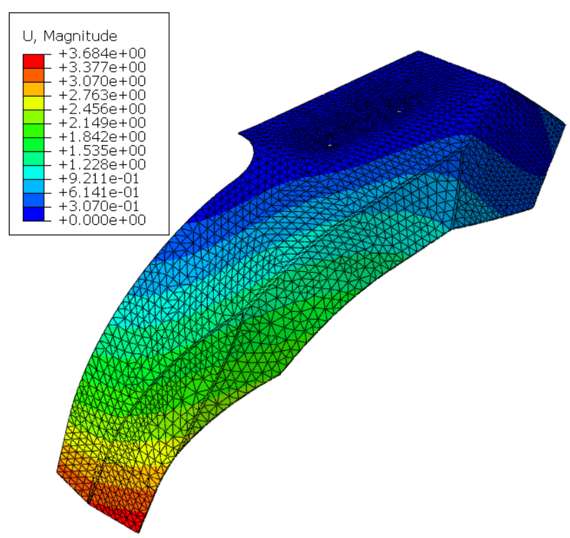


Nonlinear material data was then entered. AME requires nonlinear stress-strain data to failure for both flow and cross-flow directions (Material > Add non-linear Material). The data was used to reverse engineer the properties for nonlinear material behavior in the structural simulation, and the new Abaqus input file was exported from AME to be run in Abaqus, either locally or on a Linux cluster (Home > Export to Structural Package).

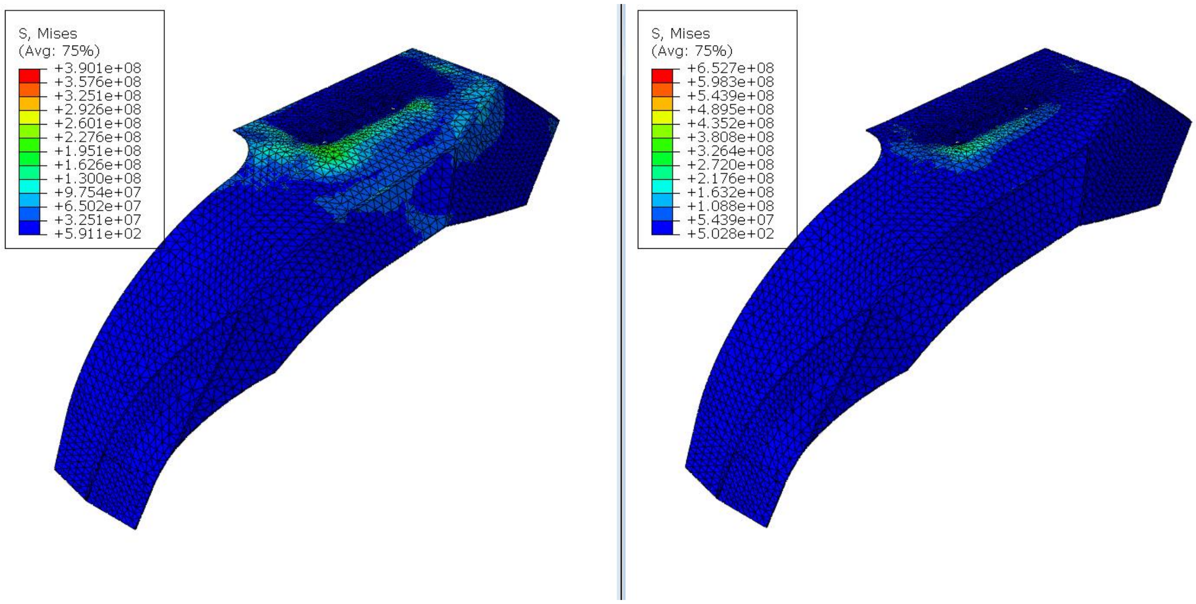


An Abaqus simulation was also ran for an isotropic, linear elastic material for the same loading. Below, we compare the displacements of the linear elastic model (right) and the nonlinear 'as-manufactured' model (left). It is evident that the material does indeed reach the yield strength and softens considerably, which leads to more deformation.





Similarly, looking at the von-Mises stress at the same point in the load history shows considerably less stress due to the softening behavior of the material from the plasticity.

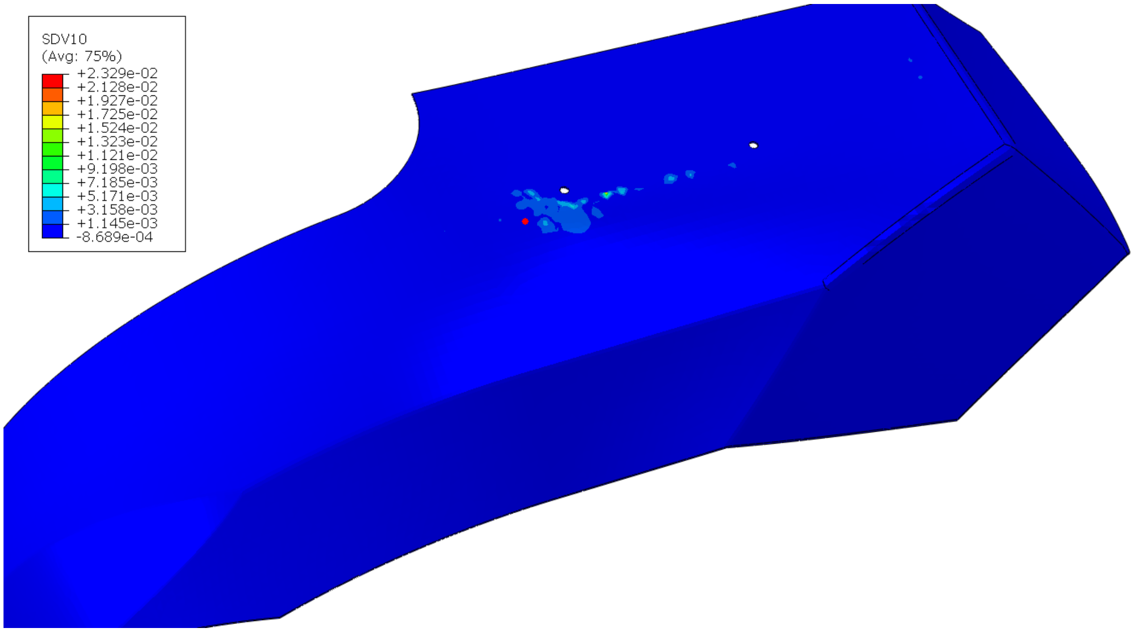


Understanding Outputs from ASCA

Relying on displacements and von-Mises stress values to determine performance works well for understanding general behaviors of the structure. More interesting, though, is predicting yielding and rupture of the structure.

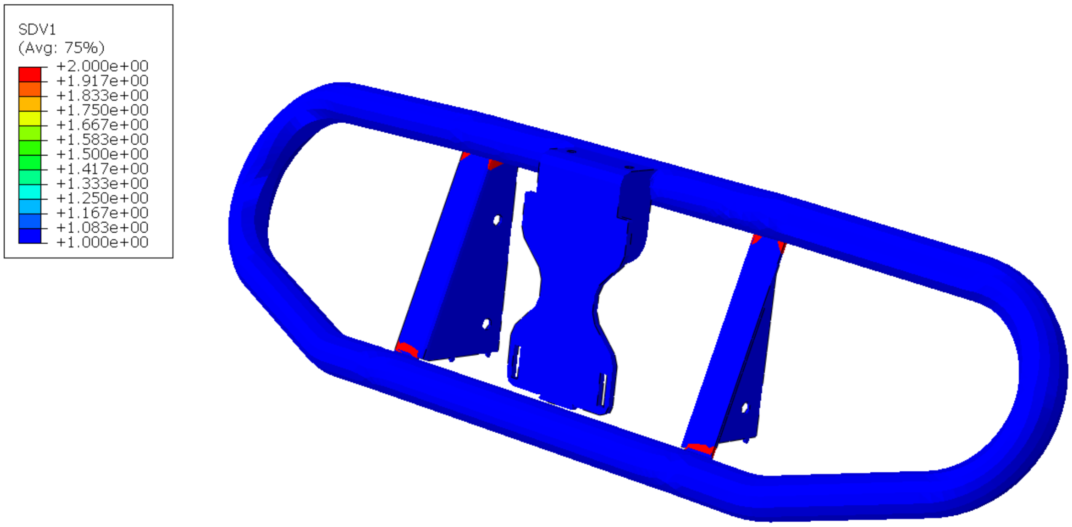
Design constraints will dictate the type of behavior which is acceptable for the structural performance. The majority of designs rely on keeping the material in the elastic regime (e.g. no plasticity). Solely relying on the von-Mises stress values does not provide sufficient insight into whether the material remains in the elastic regime. ASCA outputs measures of plasticity and rupture in the form of User-Defined State Variables (SDVs).

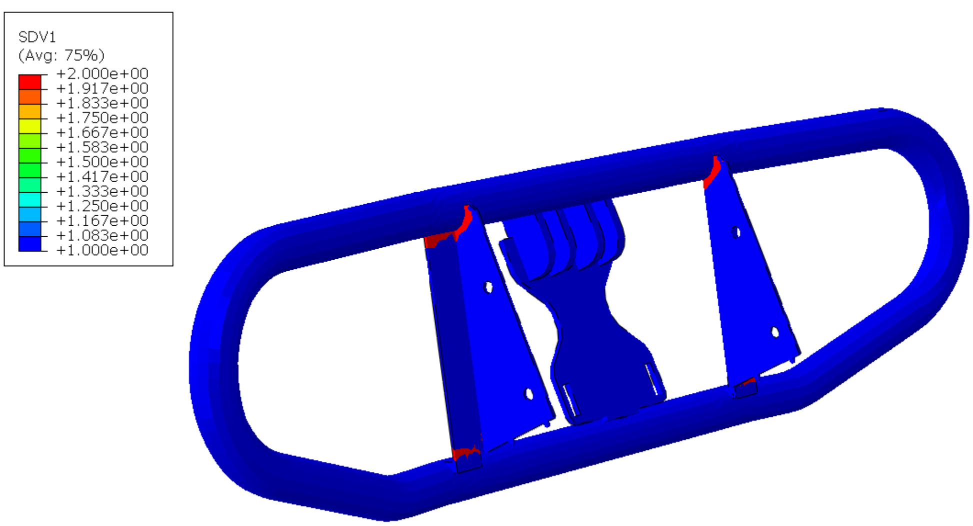
Viewing plastic strain accumulation in the matrix constituent is easily visible by requesting output of SDVs from Abaqus and viewing them in the post-processor. Below is a picture of the plastic strain accumulation in the matrix constituent (viewing SDV10) at the end of the load history.



Incrementally stepping through the load history provides a description of where plastic strains initiate and how they evolve, allowing the geometry, material, or manufacturing process to be altered to meet specifications early in the design process.

ASCA also outputs failure modes of the composite material as SDV1. If matric rupture is predicted, ASCA will degrade the integration point material properties appropriately by lowering the stiffness to a near-zero value. This will redistribute loading to surround integration points (and elements) and allow damage to propagate. Investigating SDV1 will inform users if damage is present or not, where it begins, and how it evolves. Values of 1.0 indicate no failure, and values of 2.0 indicate matrix failure.


SDV1
(Avg: 75%)
+2.000e+00
+1.917e+00
+1.833e+00
+1.750e+00
+1.667e+00
+1.583e+00
+1.500e+00
+1.417e+00
+1.333e+00
+1.250e+00
+1.167e+00
+1.083e+00
+1.000e+00


SDV1
(Avg: 75%)
+2.000e+00
+1.917e+00
+1.833e+00
+1.750e+00
+1.667e+00
+1.583e+00
+1.500e+00
+1.417e+00
+1.333e+00
+1.250e+00
+1.167e+00
+1.083e+00
+1.000e+00