AUTODESK UNIVERSITY 2015

DE10212

Autodesk Helius PFA: Advanced Material Simulation for Light-Weighting

Dan Milligan
Autodesk

Doug Kenik
Autodesk

Learning Objectives

- Learn how to identify use cases and the benefits for using advanced material simulation tools
- Learn best practices for using Helius PFA
- Discover the newest features available in Helius PFA 2016
- Discover the future of advanced material simulation at Autodesk

Description

Plastics, composites, and other advanced materials are replacing metal across multiple industries. The main benefit provided by replacing metallic designs with advanced material designs is the ability to make a product lighter. Advanced materials add challenges to the design process for parts, assemblies, and structures that use these materials. The material behavior is often non-linear, anisotropic, and influenced by how products are manufactured. Helius PFA software provides powerful capabilities for simulating these beneficial but complex materials within a mechanical design. The Helius PFA software suite includes Advanced Material Exchange software for mapping manufacturing data from Moldflow Insight to structural simulation. Join Autodesk, Inc., to learn about the newest features, use cases, best practices for using Helius PFA software, and on the future of advanced material simulation at Autodesk.

Your AU Experts

Dan Milligan *is an engineer with Autodesk, Inc., and previously Firehole Composites. Dan has worked in the advanced materials industry since graduating from the University of Wyoming. He focuses on using simulation software for the design and analysis of products made using advanced materials. Dan has presented at Autodesk University and many advanced material conferences, such as SAMPE, the Composites and Advanced Materials Expo (CAMx), JEC, and NPE.*

Doug Kenik *is a product line manager for composite simulation products within Autodesk, Inc. He holds both an MS and a BS in mechanical engineering from the University of Wyoming, where he spent his graduate career developing high-fidelity micromechanics models for composite material simulation. Prior to working at Autodesk, Doug spent 5 years as a developer and application engineer at Firehole Composites, where he helped implement new technologies for composite simulation and define next-generation enhancements for use within existing products.*

Benefits of Using Advanced Material Simulation Tools

Advanced materials such as plastics and composites are shaping how we design and manufacture products. One of the first places we see advanced materials creating new trends in design and manufacturing is in innovation. Manufactures strive to continually innovate with the design of their products. Innovation will give them a competitive edge against other companies, or even better, disrupt the market and create an entirely new category of products that are uniquely positioned to satisfy consumer demand. We see two examples of this in Figure 1 and Figure 2. Figure 1 depicts the increase in popularity of drones that are manufactured using fiber reinforced plastics (FRPs) because of the requirement of being lightweight in order to increase payload capacity while also being strong to avoid breaking in the event of a crash. Figure 2 depicts the Mercedes Biome Concept Car that is using FRPs to lightweight both structural and non-structural components in order to achieve optimal fuel economy and performance.

FIGURE 1: DRONE MANUFACTURED USING FRPS

FIGURE 2: MERCEDES BIOME CONCEPT CAR

In the automotive industry, regulations and consumer demand for vehicles with increased fuel economy are accelerating. In the United States, there is a mandate for the fleet average fuel economy of new vehicles to be 54.5 MPG by the year 2025. With different options available to achieve these fuel economy targets, automotive light-weighting through the use of advanced materials, such as plastics and composites, is emerging as a clear favorite.

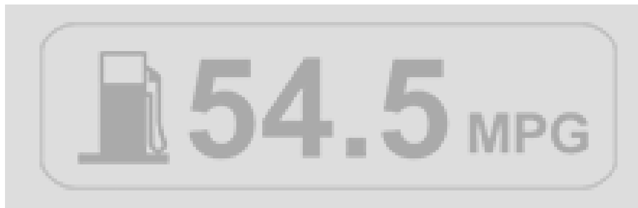

FIGURE 3: CAFE STANDARD FOR FLEET AVERAGED FUEL ECONOMY BY 2025

A persistent trend is to start using more plastic materials in structural applications. Fiber filled plastics have long been used for structural applications, but there is an increased move toward using fiber reinforced plastics to produce even stronger lightweight parts. Once fibers are added to a base plastic, we now have a composite material (Figure 4).

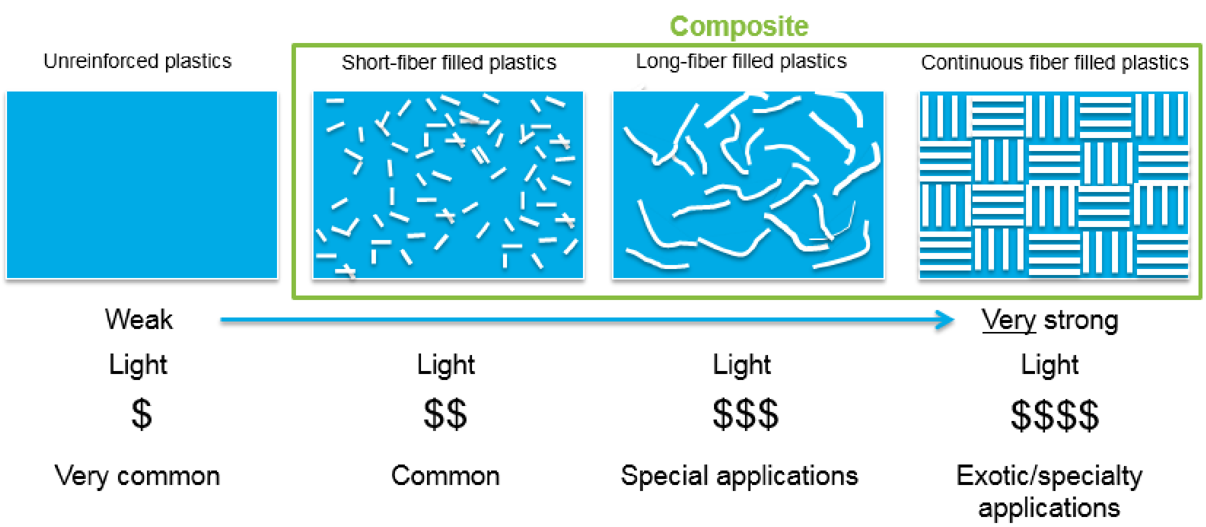

FIGURE 4: COMPOSITE MATERIALS ARE FORMED BY ADDING FIBERS TO UNREINFORCED PLASTIC MATERIALS

As you can see in Figure 4, as we add longer and longer fibers to the base plastic material, 2 things happen:

1. The strength of the material increases.
2. The price of the material increases – this leads to the fiber filled plastics being used on a smaller subset of parts and forces designers and engineers to get the design right the first time to minimize scrap.

There are many workflows that we typically see when it comes to reinforced plastic part design. The most common is that a CAD model is used separately by a team for manufacturing simulation and also structural simulation as shown in Figure 5. Each team runs the CAD model through a preprocessor and performs their own simulation. However, the manufacturing and structural simulations are disconnected, running in parallel. There are a couple of potential problems with this:

1. Both simulation teams could be changing the geometry independently to meet the design needs and criteria.
2. Structural simulation usually approximates the material as isotropic, not taking into account how the fiber orientations affect the stiffness of the part.

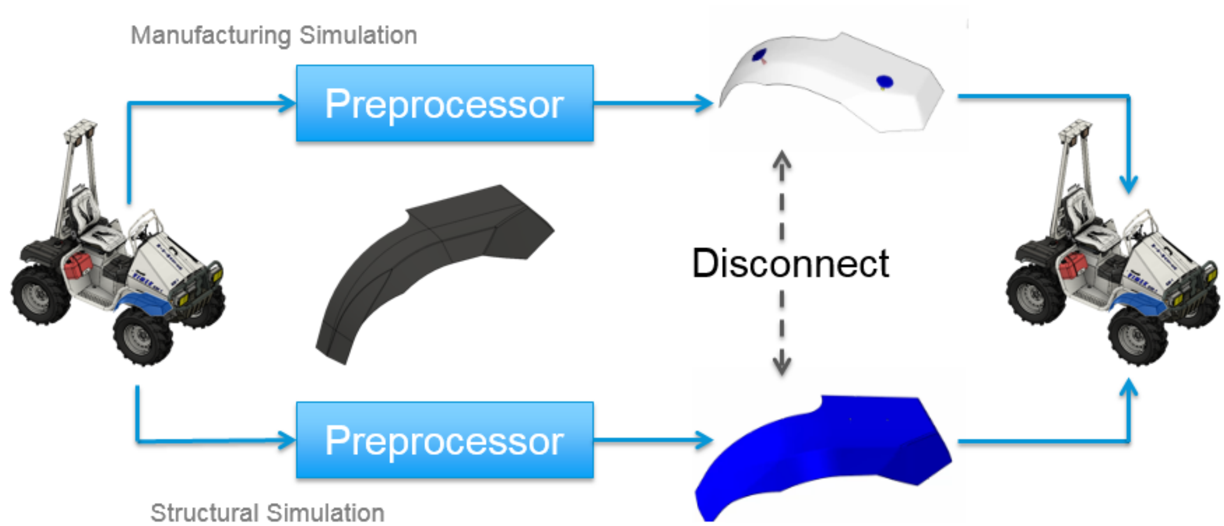

FIGURE 5: PARALLEL DESIGN PATHS BETWEEN MANUFACTURING SIMULATION AND STRUCTURAL SIMULATION WITHOUT THE ABILITY TO SHARE INFORMATION

Accounting for Orthotropic Material Behavior
Looking first at the manufacturing simulation path, the first place that Autodesk software can help you get better simulation results is by using Autodesk Moldflow to simulate the fiber orientations of a chopped fiber reinforced plastic when it is injection molded. Figure 6 shows an example of this with a plaque that is injection molded on the left hand side. The mold filling progress results in a few different regions of fiber orientations:

1. Expanding flow near the injection gate results in a preferred fiber alignment in the “X” direction.
2. As the expanding flow front hits the edges of the mold and the central cutout, a more random orientation occurs.
3. As the flow front starts to move uniformly past the central cutout along the wall edges, the fibers become highly aligned again in the “X” direction.

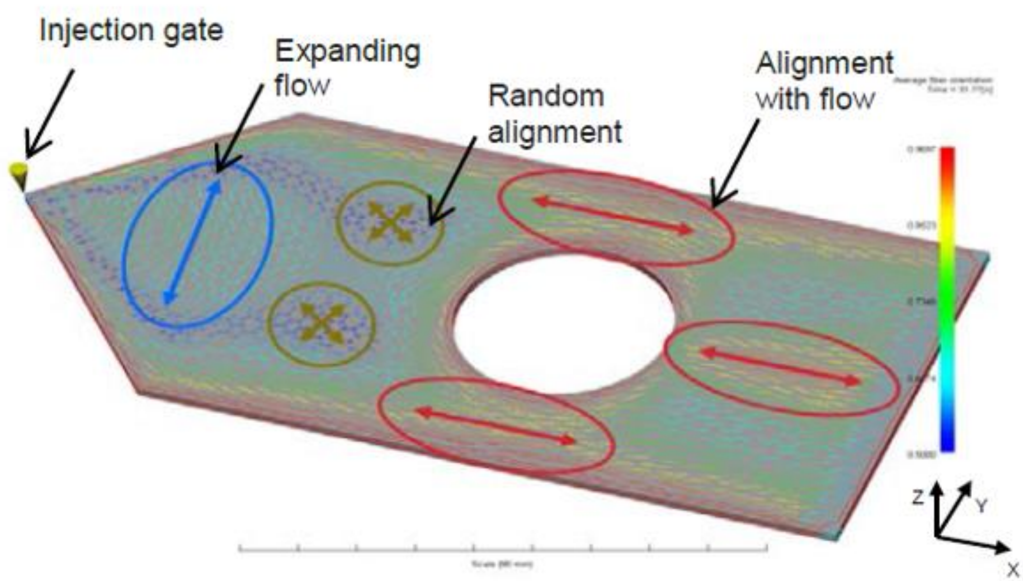

FIGURE 6: FIBER ORIENTATIONS THAT RESULT FROM AN INJECTION MOLDING PROCESS

Moldflow can predict the fiber orientations from an injection molding process, by why are fiber orientations important? As you can see in Figure 7, once you add fibers to a plastic material, you transform the plastic material from being an isotropic material that has the same stiffness in any direction, into an anisotropic material where the stiffness of the material is different depending on the orientation of the fibers.

If the fiber are highly aligned, like the image on the left, you can see that the stiffness in the X direction is 3 times higher than the stiffness in the Y direction.

If the fiber are more randomly aligned, the material behaves more like an isotropic material with equal stiffness in the X and Y direction.

And these are just two cases. There are infinite possibilities of in-between behavior depending on how much the fibers are aligned and in which direction. It is this exact type of information that we can get out of an Autodesk Moldflow injection molding simulation.

	Fibers in X direction	Random Alignment
Ex (MPa)	12000	5500
Ey (MPa)	4000	5500

FIGURE 7: STIFFNESS BEHAVIOR OF FRPS ARE INFLUENCED BY FIBER ORIENTATION

Going back to our example plaque, now if we look at the stiffness in the X direction we can see how fiber alignment affects the stiffness of the material (Figure 8). If we were to assume isotropic material behavior, we would not be able to capture this spatially varying stiffness.

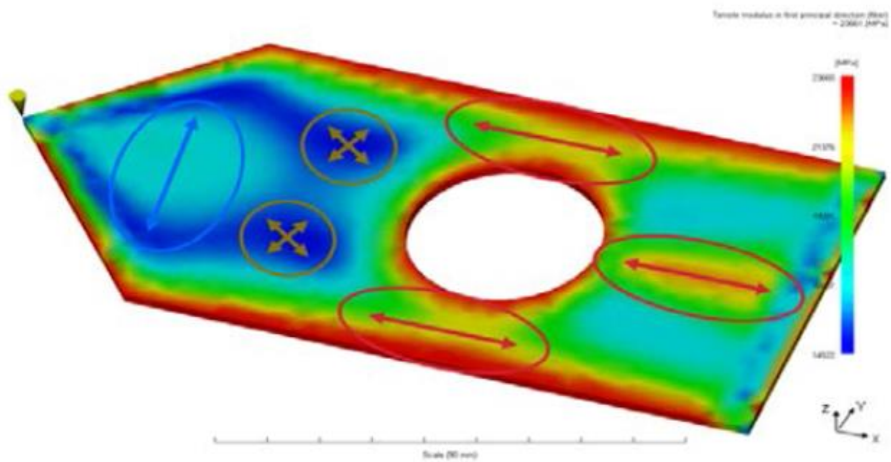

FIGURE 8: STIFFNESS IN X DIRECTION VARIATION DUE TO FIBER ORIENTATION

Strength/Failure Predictions
In addition to accounting for stiffness variation in an injection molded part, fiber orientations will also have a large influence on the strength of a part. Consider the example shown in Figure 9, where we are cutting a biaxial coupon from an injection molded part.

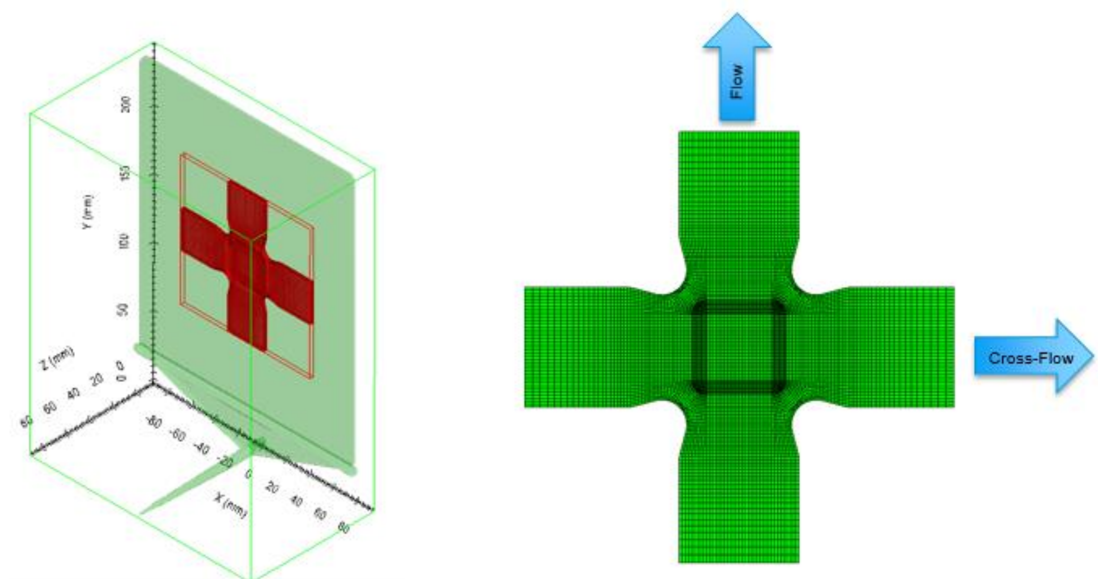

FIGURE 9: BIAXIAL COUPON CUT FROM INJECTION MOLDED PLAQUE

The preferred flow direction in the vertical (Y) direction results in the majority of the fibers aligning in the Y direction. This will present a scenario where not only will the stiffness value be different in the X and Y directions as we have seen previously, but the strength value will vary significantly also. Experimental results for the biaxial coupon tested in various directions are shown in Figure 10.

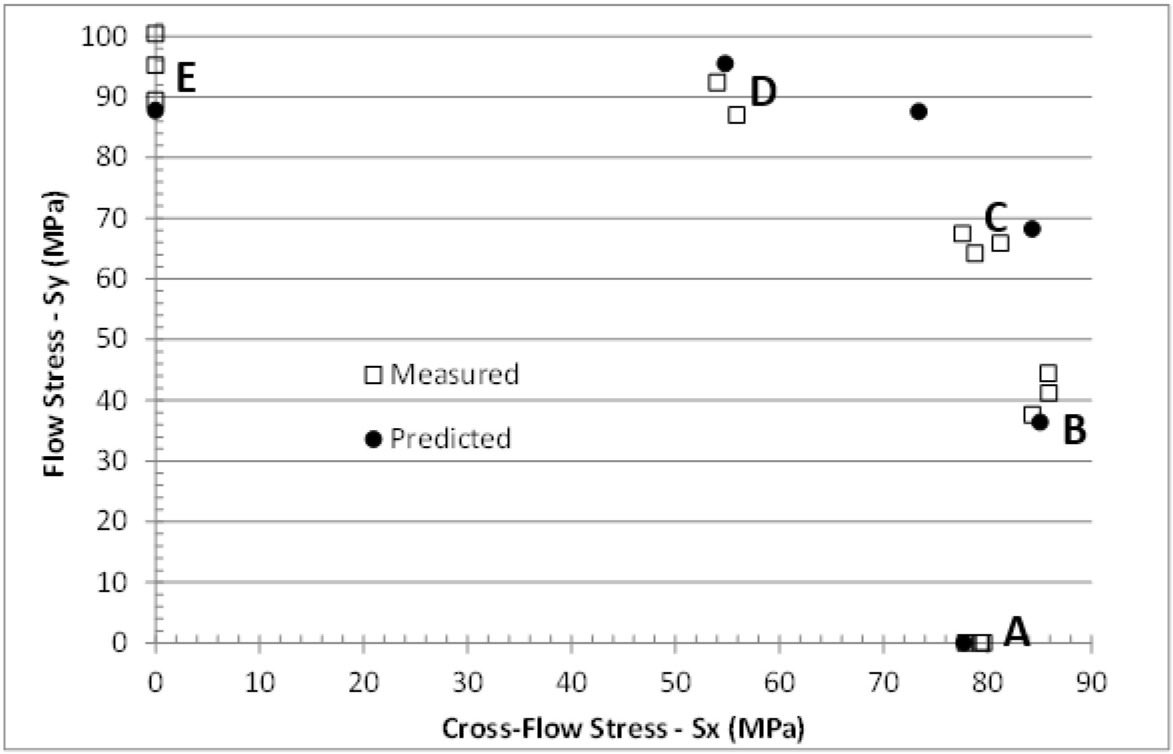

FIGURE 10: EXPERIMENTAL AND SIMULATED STRENGTH VALUES FOR A BIAXIALLY LOADED COUPON TESTED UNDER DIFFERENT LOAD RATIOS

What you can see in the plot in Figure 10 is that the biaxial coupon is failing at a stress level of 80 MPa when loaded in the cross-flow (X) direction and at a higher value of 95 MPa when loaded in the flow (Y) direction. This is because the fiber alignment in the Y direction increases the strength of the coupon due to the reinforcing behavior of the fibers. This difference becomes even more apparent as the length of the reinforcing fibers increases.

Examples of light-weighting using Moldflow and Helius PFA

Now that we have discussed the benefits of accounting for fiber orientations, let's a take a look at how to use the technology to help engineer lightweight components.

Modifying Gate Locations to Optimize Clearance after Assembly

One of the most critical aspects of injection molded components for exterior automotive components is the clearance of the component after it has been assembled to the structure. After the component is ejected from the mold, it will warp. There are many factors driving the warp magnitude and direction, but one of the most prominent is the fiber orientation of the component.

After the component has cooled, it is assembled to the larger structure. If the component has warped enough, it can have a significant impact on the 'lines' of the car after it has been assembled. This is due to the residual strains in the component driving warpage. A warped component (or part) will have to be pressed and secured into location, which will induce stresses and strains in a warped component. To illustrate this, consider the following scenario

A fender for a car must be able to take a decent amount of load without buckling, cracking, or going into a plastic regime. One might consider using a fiber filled thermoplastic for the fender. After the part is ejected from the mold, the pat warps on the table and then is assembled onto the car once cooled. When the fender is being assembled, it has to be 'pressed' into place and then secured because the warpage has cause the component to deflect from its designed geometry. We want to be sure that the component does not affect the lines of the car, and that the clearance between other assembled components is not enough to cause conflict.

A picture of the assembled component in question is shown in Figure 11.

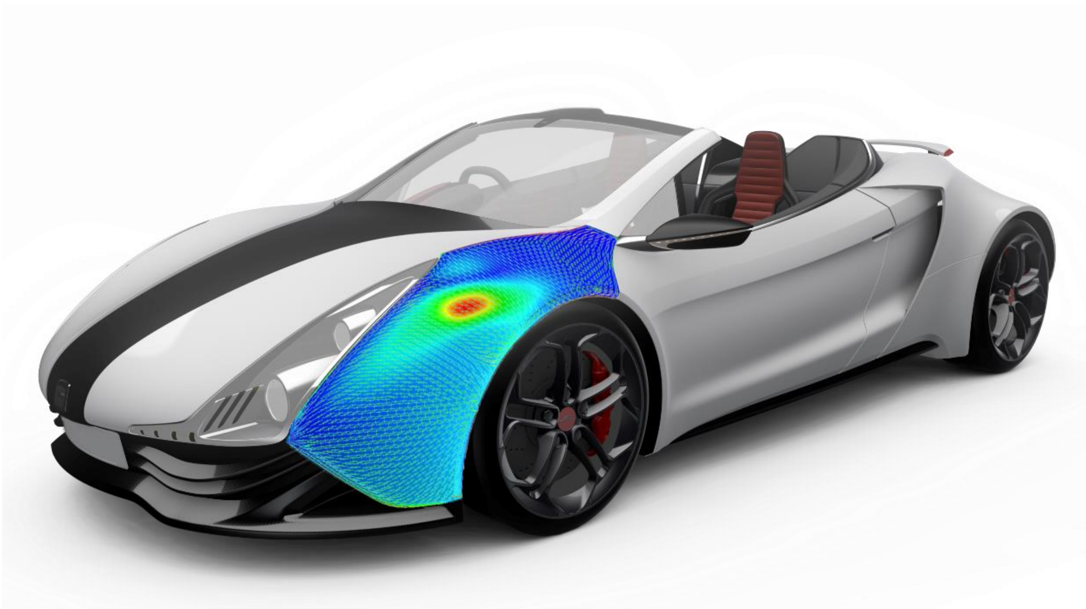

FIGURE 11: INJECTION MOLDED FRONT FENDER

A picture of the component is shown Figure 12, with mounting locations exposed.

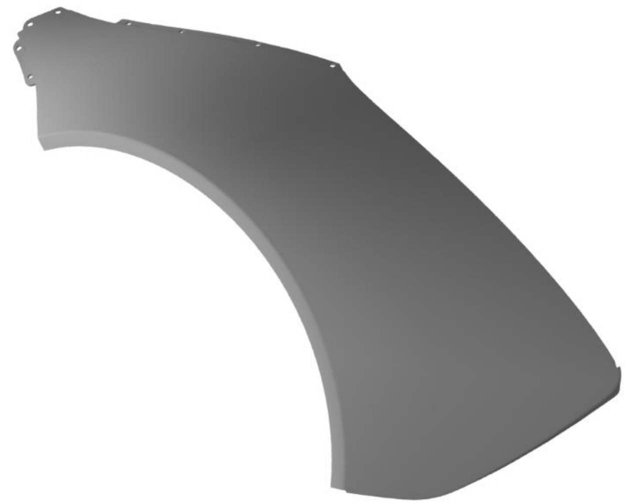

FIGURE 12: MOUNTING LOCATIONS ON FRONT FENDER

Now follow along as we perform the following:

1. Run Autodesk Moldflow to determine the fiber orientation and warpage of the component
2. Use Advanced Material Exchange to map fiber orientations and warpage from Moldflow to structural simulation
3. Simulate the deformation of the component after it has been manufactured

We will compare multiple gate locations in the Moldflow simulation and compare the final deformed shape after assembly.

Determining the Natural Frequencies of a Plastic Bracket

A very common use case for light-weighting is to replace metal brackets with plastic brackets. A typical car has many brackets. Substituting plastic brackets for metal brackets can save a lot of weight. For example, let's assume a typical steel bracket weighs approximately 75 grams. By simply substituting plastic (without redesigning for structural integrity) one can reduce the weight to 15 grams, which is a 5x improvement in weight just for the bracket. If 50 brackets are substituted with plastic, one could hypothetically save 3 kilograms, or 6 pounds in weight. It might not seem like a lot, but these types of substitutions add up.

Brackets are typically designed for multiple load cases. Factors that are critical include strength, fatigue, and the natural frequency. In this example we will look at the natural frequency of the bracket. Ensuring brackets do not fall within the natural frequency of the car is paramount to their design. If the natural frequency of the bracket lies within the natural frequency of the car, the component will likely fail due to the excitation frequencies matching and amplifying the loads on the bracket. The natural frequency of the bracket depends on the geometry and material description. The fiber orientations of the material can have a sizeable effect on the natural frequency of the component.

As an example, let's take a look at the natural frequency of the bracket shown in Figure 13.

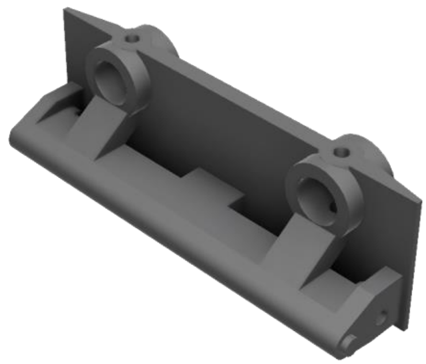

FIGURE 13: INJECTION MOLDED BRACKET

For this problem, we will perform the following steps:

1. Run the manufacturing simulation using Autodesk Moldflow
2. Transfer the fiber orientations from Moldflow to Abaqus using Advanced Material Exchange
3. Run a frequency analysis in Abaqus to determine the first 5 natural frequencies and their modes

To make this a bit more interesting, we will also compute the natural frequencies of the bracket by assuming the minimum (cross flow) and maximum (flow) material properties for the bracket and compare these results to those for the as-manufactured simulation.

Advanced Material Simulation Best Practices

In order to account for how fiber orientations will affect the behavior of injection molded parts there are multiple best practices that are recommended in order to get the best results from your simulation:

1. Account for fiber orientations
2. Use non-linear test data
3. Improve the convergence behavior of your FEA model

Account for Fiber Orientation

Knowing that Autodesk Moldflow can predict fiber orientations, now we can account for the fiber orientations in our structural simulation. Autodesk Advanced Material Exchange can bridge the gap between manufacturing simulation and structural simulation by mapping material properties and fiber orientations from an Autodesk Moldflow simulation to a structural simulation (Figure 14). Then Autodesk Helius PFA uses this information to run a FEA analysis that accounts for the as manufactured properties of the material.

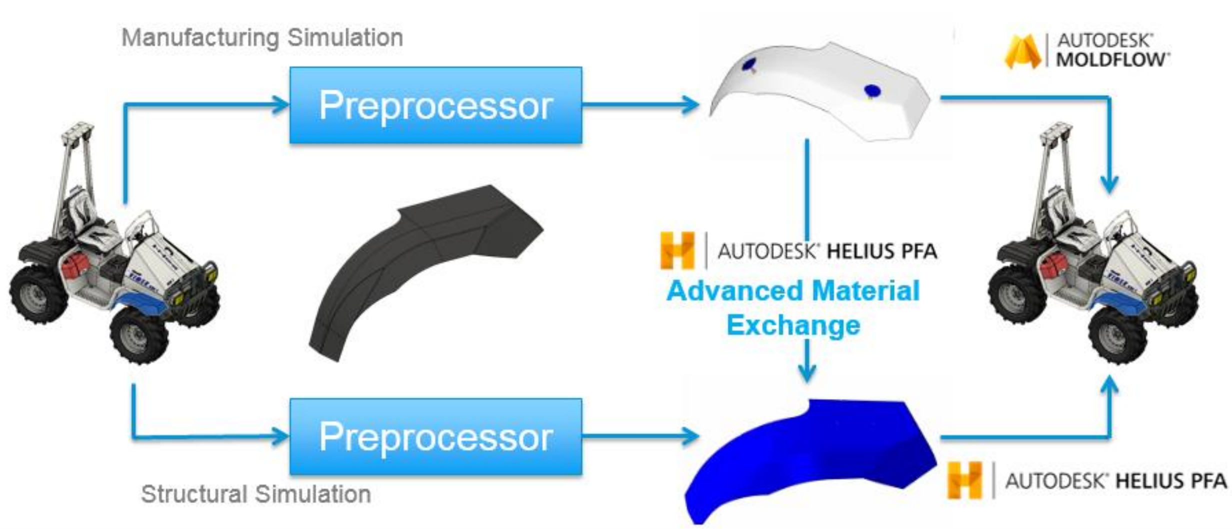

FIGURE 14: AUTODESK ADVANCED MATERIAL EXCHANGE ALLOWS YOU TO MAP FIBER ORIENTATIONS FROM A MOLDFLOW SIMULATION TO A STRUCTURAL SIMULATION

This allows the structural simulation to account for that variation in material behavior that we saw due to the injection molding process and how fibers are oriented in the part. The key advantage with the Advanced Material Exchange software is that different geometries, element types, mesh densities can be used and the software will automatically map this data from model to the other. Figure 15 shows an example of fiber orientations being transferred from a Moldflow model to a FEA model using the Advanced Material Exchange interface.

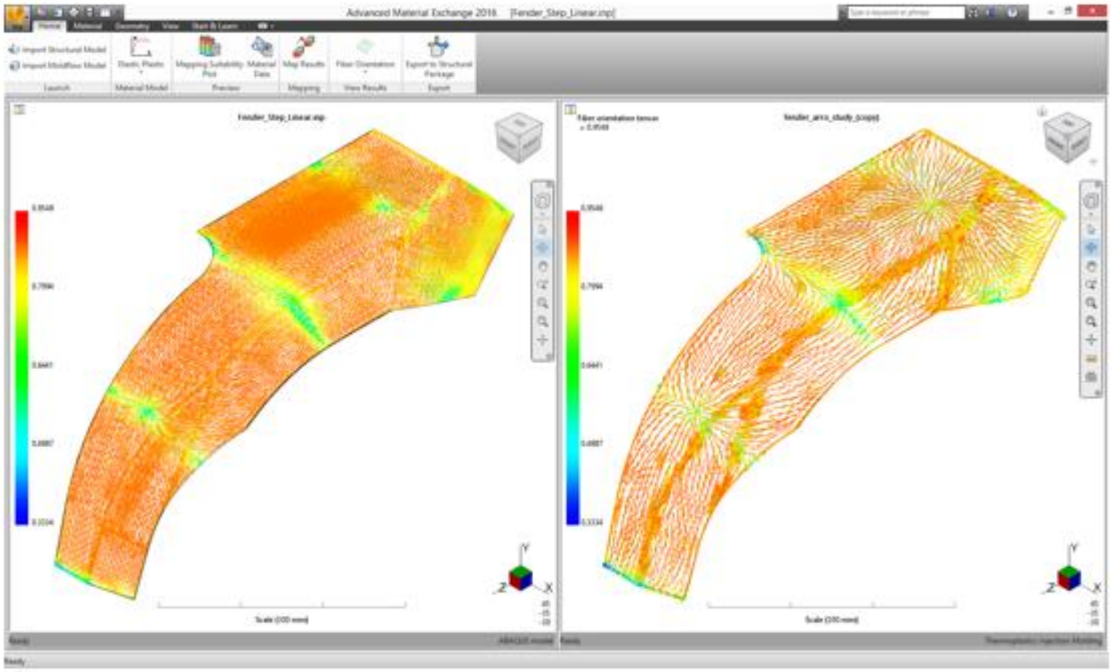

FIGURE 15: AUTODESK ADVANCED MATERIAL EXCHANGE INTERFACE FOR MAPPING FIBER ORIENTATIONS FROM MOLDFLOW TO A FEA MODEL

Use Non-Linear Test Data

A second best practice for getting the best results from your structural simulation is to account for nonlinear material behavior. A nonlinear stress-strain curve for a typical FRP is shown in Figure 16. Capturing nonlinear material behavior might not be critical at low load levels, but as you can see in Figure 16, once you reach higher levels of stress and strain in our FEA model, FRPs start to behave in a very nonlinear fashion and accurate structural results require us to model this nonlinear material behavior.

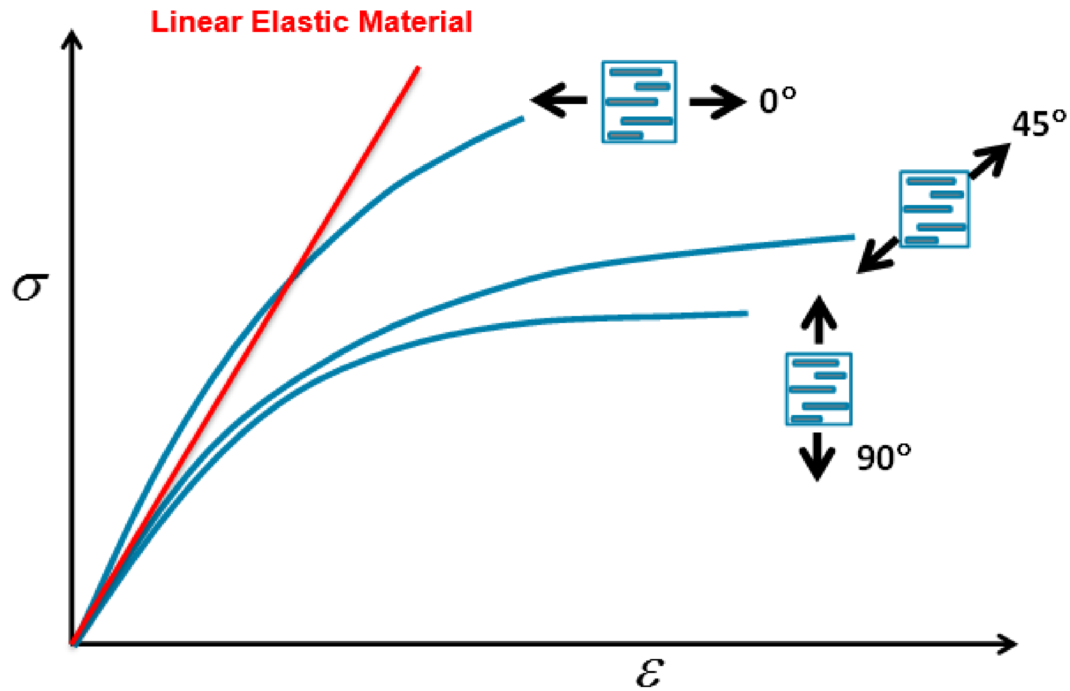

FIGURE 16: NONLINEAR MATERIAL BEHAVIOR OF FRP MATERIALS

When we want to run a nonlinear material simulation, we have the ability to add nonlinear stress-strain curves into the Advanced Material Exchange software. There are 3 stress-strain curves that are required:

1. Flow Direction (0°)
2. Cross Flow Direction (90°)
3. 45° to Flow Direction

An additional advantage that the Advanced Material Exchange tool provides is that test data can (and should) be entered all the way out to failure or rupture of the material, not just yield. This allows you to model the nonlinear behavior of the material up to and past rupture of the material so that we can simulate the rupture of the FRP in our FEA model. This is important for determining the strength of parts made from FRPs.

Improve Convergence Behavior
Modeling the damage due to rupture of a FRP material involves a modeling process called progressive failure analysis (PFA). A progressive failure analysis:

1. Identifies when failure occurs at a point in the FEA model

And

2. Reduces the stiffness of the material at that point to replicate a weakened material due to failure

With this process, you can simulate how damage spreads through a part/structure as loading increases (the progression of failure). A progressive failure analysis results in highly non-linear material behavior of the FRP, so there are some recommendations for how to set-up a model for a progressive failure analysis.

Minimize amount of nonlinearity

There are 3 main types of nonlinearity in a FEA model:

1. Material (nonlinear material behavior)
2. Geometric (large displacement theory)
3. Contact (evolving interaction between parts)

The best convergence behavior occurs when you can eliminate (2) and (3) from above. (2) can be turned on or off inside the FEA software. (3) can be eliminated by using boundary conditions and loads instead of contact to load a particular component.

Divide the analysis up into small time increments

To take advantage of a progressive failure analysis, the analysis needs to be divided into multiple increments. This allows you to examine how damage progresses through a structure. It is recommended that the analysis be divided into 20-100 increments. Fewer increments will result in a faster runtime while more increments will provide better resolution of the results.

Use displacements instead of forces

Loading a FEA model by imposed displacements instead of imposed forces is recommended when performing a progressive failure analysis. This is because as failure spreads, the model becomes more and more compliant. If the model was loaded by forces, the forces would keep moving higher even after the part has catastrophically failed which can result in erroneously large displacements that will cause convergence problems. When displacements are used, the force required to impose the displacement will decrease as failure accumulates which keeps the model in a more realistic and computationally effective state.

Discover the newest features available in Helius PFA 2016

Helius PFA is a 'Term' product, which means that users rent the use of the product in annual increments. The benefit to having term products is that updates to the product are scheduled regularly, and the team is allowed to deliver new functionality whenever it is available. The other large benefit to our customers is that they are automatically given the new versions and the included support to ensure success.

To that end, we recently released an update to Helius PFA in October 2015 to provide new functionality to our users. We have been working with current and potential customers, and we were able to identify areas which could return maximum value. Our goal for the October release was 3 fold:

1. Focus on improving our current workflows
2. Set the stage for the next major release
3. Deliver Beta technology to customers interested in trying and providing feedback

The product will continue to deliver Beta technology which the team is requesting feedback on. This allows the product to continue to grow and deliver cutting edge solutions to very difficult problems, and also provide visibility to the customer base on how we are solving some of the more challenging problems.

The latest release had a large focus on updates for new technology within Advanced Material Exchange, including:

Support for all elements

Previous releases of AME would only support specific element types in an Abaqus or ANSYS input file. In dealing with our customer base, it became apparent that there could be multiple element types in a structural input file, including:

- Multiple solid or shell higher order elements
    - 20 node bricks
    - Wedges
    - Etc.
- Rigid surfaces
- Contact elements
- Analytical surfaces
- Spring elements
- Etc.

AME was modified to accept all of these various element types. Now users can read in a structural input file with multiple parts/materials and any element type. If AME cannot map to a specific element type (like a spring element or rigid surface) it will warn the user and map to those element types supported.

Mapping Updates

AME now also supports mapping for the following:

- Thermoplastic injection molding
- Thermoplastic injection-compression molding

Material Support

We have also included new capability for specific material types. AME currently supports both linear and elastic-plastic (nonlinear) material responses. Choosing one over the other depends on multiple factors. In general, linear material responses are great for quick studies on stiffness and stress, and elastic-plastic responses are ideal for determining if any plasticity or rupture will occur during the life of the component.

For linear material responses, AME will map over all of the material data from Moldflow to structural, including CTE's. The user does not need to enter any further data. For nonlinear material responses, AME will map over all of the data as well which is included with Moldflow. If nonlinear data is not in the Moldflow material database for the chosen material, that data must be included in AME.

AME also supports fiber filled, talc filled, mineral filled, and unfilled material responses. Any filler with an aspect ratio > 1 is considered to be fiber filled, and the resulting material properties are anisotropic and dependent on the orientation of the filler.

Any material with a filler which has an aspect ratio = 1 (mineral and talc filled) are considered to be isotropic, even if the data suggests they may be orthotropic. AME will take the maximum elastic modulus and Poisson ratio for the material. This is the same for orthotropic unfilled materials. In general, orthotropic unfilled, talc filled, or mineral filled materials do not exhibit a significant orthotropic response. In general, the stiffness in the 2 in plane directions are within 10% of each other, suggesting an isotropic approximation will work.

Material Fitting

Selecting elastic-plastic material response requires the user to have nonlinear stress-strain data for the material. If this data is not already present in the Moldflow material database, the user must enter it in AME so it can be characterized for the material model. Prior versions of AME had many restrictions on the input material data, including:

- The data had to have increasing stress values (e.g. it had to be smoothed)
- AME could only accept 15-50 data points per nonlinear curve

Most of the data users have from material suppliers or 3rd party mechanical tests do not completely satisfy these requirements. This then forces the user to smooth the data and reduce it to a feasible size to work with AME.

The new release of AME removes these restrictions. Now users can enter any amount of nonlinear data, and the data does not have to be smoothed. AME will smooth the data for the user and sample the data at various locations determined stochastically from the input data. This relaxation ensures characterizing materials is as easy as possible for our users.

New Mechanical Tests and Material Fitting

Finally, and arguably most importantly, Autodesk now offers mechanical testing for AME. This is significant for a variety of reasons:

- Now customers can have their material tested for **both** Moldflow and Helius PFA with AME
- Autodesk will mechanically test the material and use the data to characterize the nonlinear response of the material
- Autodesk will deliver the user a new material file (.udb) to be included in the user's Moldflow material database
- Whenever a user brings in a Moldflow Study using the characterized material, the nonlinear data and associated fit is automatically brought into AME and used in Helius. The user does not need to enter the nonlinear data!

Now users can populate their material database with the nonlinear data and use that data in Moldflow and Helius PFA. The nonlinear data does not need to be entered in AME, which allows the user to skip the most tedious step in the AME workflow and reduce the amount of time required for mapping.

Discover the future of advanced material simulation at Autodesk

While we cannot disclose this in the class handout, please come to the class to discover what we are doing for the future of advanced materials, including:

- New enhancements for AME
- Discover the future of advanced material simulation at Autodesk
- New material models for continuous fiber composites
- New workflows for continuous fiber composites