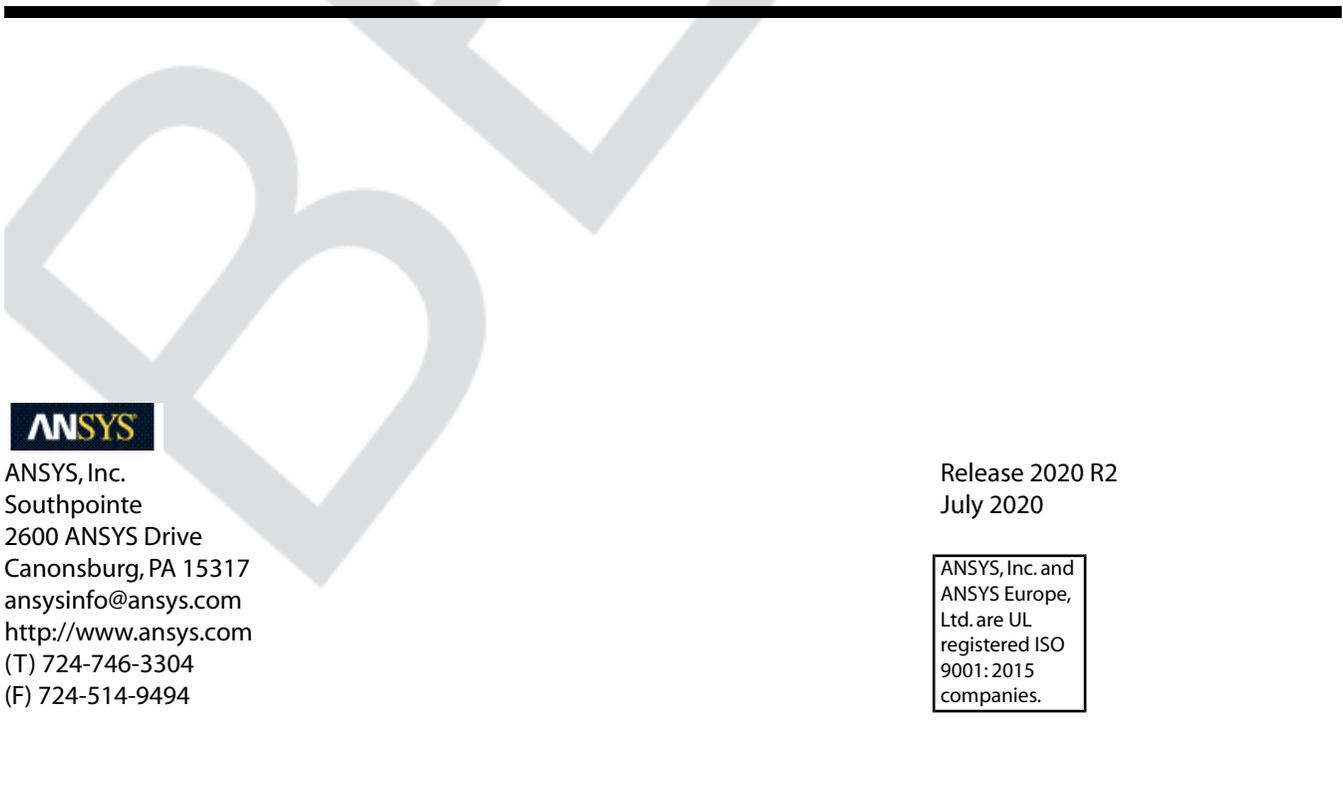


Short Fiber Reinforced Composites Simulation in Workbench (Beta)



ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
<http://www.ansys.com>
(T) 724-746-3304
(F) 724-514-9494

Release 2020 R2
July 2020

ANSYS, Inc. and
ANSYS Europe,
Ltd. are UL
registered ISO
9001:2015
companies.

Copyright and Trademark Information

© 2020 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXIm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

DISCLAIMER

This document describes beta features.

Beta documentation and features are considered unreleased and have not been fully tested nor fully validated. The results are not guaranteed by ANSYS, Inc. (ANSYS) to be correct. You assume the risk of using beta features and beta documentation.

At its discretion, ANSYS, Inc. (ANSYS) may release, change, or withdraw beta features in future revisions.

Beta features are not subject to the ANSYS Class 3 error reporting system. ANSYS makes no commitment to resolve defects reported against beta features; however, your feedback will help us improve the quality of the product.

ANSYS does not guarantee that input files used with beta features will run successfully from version to version of the software, nor with the final released version of the features. You may need to modify the input files before running them on other versions.

Table of Contents

1. Short Fiber Reinforced Composites Simulation in Workbench	1
1.1. Introduction to Short Fiber Reinforced Composites	1
1.1.1. How Simulation Can Assist	1
1.2. Simulating Short Fiber Reinforced Composites in Workbench	2
1.2.1. Methodology	3
1.3. Constituents Material Data	4
1.4. Microscale Homogenization in Material Designer	5
1.5. Import of Injection Molding Simulation Results	7
1.6. Setting Up the Mechanical System	8
1.6.1. Import Element Orientations	8
1.6.2. Import Material Fields	9
1.6.2.1. Visualize the Homogenized Material Properties	10
1.6.3. Import Initial Stress	10
1.6.4. Solving	11
1.6.5. Review Results	11

BETA

List of Figures

1.1. Sampling of the orientation tensor space 6



BETA

Chapter 1: Short Fiber Reinforced Composites Simulation in Workbench

1.1. Introduction to Short Fiber Reinforced Composites

Short fiber reinforced composites are usually obtained by filling a thermoplastic or thermoset polymer with, for example, glass or carbon fibers typically less than one or two millimeters long. When compared to unreinforced polymer, short fiber reinforced composites offer improved mechanical performance while retaining the ability to be molded into complex shapes. Because of their superior properties, short fiber reinforced composites find application in the production of automotive parts, electronics, medical devices, and other industries. In fact, product designers can tune the properties of the composite by using different polymers and fiber materials as well as varying the length and amount of fibers.

Injection molding is one of the most common processes employed in the manufacturing of short fiber composite parts. Typically, the fiber reinforced polymer is heated and injected at high pressure into a mold cavity where it solidifies into the desired shape. The process, however, induces a complex fiber orientation distribution which significantly affects the mechanical response of the final part. Also, residual thermal stresses often develop due to the high temperature and injection pressure, leading to part shrinkage and warpage after demolding.

Predicting these kinds of mechanical effects is critical in product design, leaving the engineer with two options: He can resort to physical testing, which is both time consuming and expensive. Or he can employ the power of computer simulation.

1.1.1. How Simulation Can Assist

When an engineer is working with short fiber reinforced composites, computer simulation allows him to predict the mechanical properties of parts early in the design phase. The simulation must account for the following:

1. The composite microstructure and the mechanical properties of the individual constituents (polymer and fibers)
2. The manufacturing process

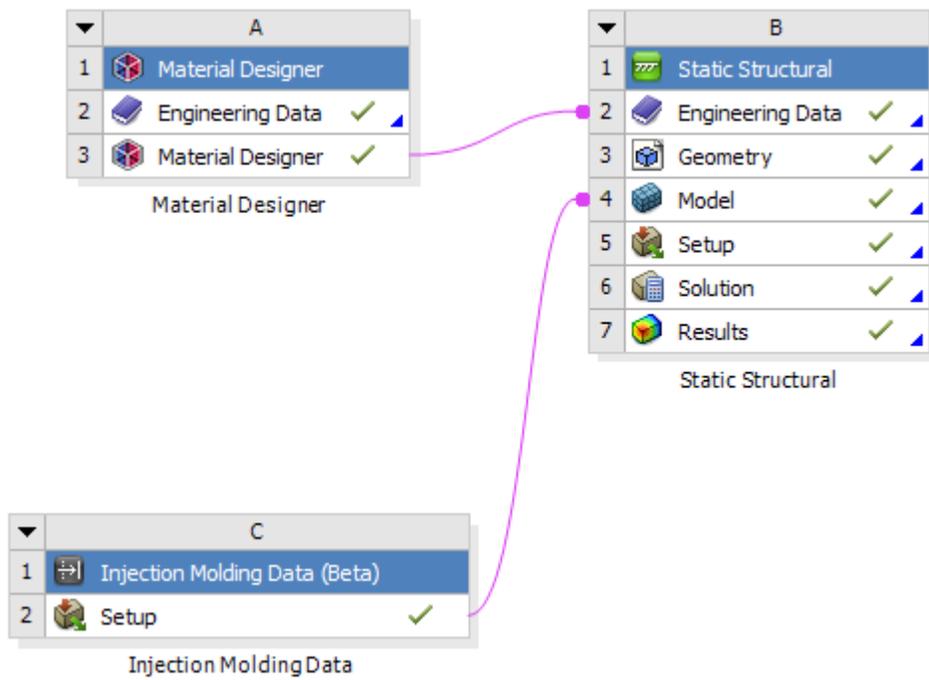
For an application to perform a comprehensive simulation, it must first model the effects of the manufacturing process and factor these results when calculating the properties of the mechanical model. This way, the simulated structure accounts for the residual thermal stresses as well as the locally varying fiber orientation distribution. As a result, the simulation can predict the stiffness and strength properties of the final part.

Once the engineer has evaluated the simulation results, he can go back and adjust the geometry or the manufacturing process to optimize performance and compensate for process-induced distortions.

1.2. Simulating Short Fiber Reinforced Composites in Workbench

Simulating the thermo-mechanical behavior of short fiber reinforced composites poses a challenge to the materials engineer, but ANSYS Workbench provides a streamlined workflow which makes the process much more manageable. The workflow, illustrated in the figure below, consists of three parts:

- In Material Designer: Using Material Designer's homogenization technology, characterize the macroscopic response of the composite material (represented in system A below).
- Using the **Injection Molding Data** system (system C below), import injection molding simulation results from other software tools. Then use ANSYS Mechanical to map the model.
- Setup of a Mechanical analysis system for simulating the part (system B).



The building blocks of the workflow are flexible, so you can adapt them to different needs and more advanced requirements. In all cases, short fiber workflows in Workbench are based on the principle that the material characterization should be independent of the part simulation. The workflow, therefore, uses a *sequential* (as opposed to *concurrent*) homogenization approach. As a result, the outcome of the microstructure homogenization performed in Material Designer becomes a regular material stored in the **Engineering Data**. This homogenized material is later fed into the structural solver without any further linking to the microstructural system. Once the homogenization has been performed, you can use the resulting material for simulating several parts, provided they are made of the same composite material.

The next section describes in more detail the methodology underlying the workflow described above.

1.2.1. Methodology

As already mentioned, the most important factors in a short fiber reinforced composite simulation are the influences of the microstructure and that of the manufacturing process. In the case of injection molded parts, the manufacturing process induces a complex, locally varying fiber-orientation distribution which significantly affects the mechanical performance of the final part. Because the orientation distribution cannot be directly computed for an entire part, injection molding simulations usually predict only its second statistical moment, the so-called (second order) fiber orientation tensor. Several dedicated software tools exist to perform this type of simulation. Typically, their output is a fiber orientation tensor field defined either at the nodes or at the element centroids of the injection molding simulation mesh (which is usually different from the structural mesh).

The following section explains how to capture the influence of the fiber orientation tensor on the overall anisotropic properties of the composite.

The fiber orientation tensor A is a symmetric positive semi-definite 3×3 tensor with trace equal to one. Therefore, it admits a spectral decomposition of the form

$$A = V^T D V \quad (1.1)$$

where

- V is a rotation matrix whose columns v_1, v_2, v_3 represent the principal fiber directions.
- $D = \text{diag}(\lambda_1, \lambda_2, \lambda_3)$ is a diagonal matrix of eigenvalues (sorted in decreasing order) with $\lambda_3 = 1 - \lambda_1 - \lambda_2$. Moreover,

$$\lambda_1 \geq \lambda_2 \geq \frac{1}{2}(1 - \lambda_1), \quad \frac{1}{3} \leq \lambda_1 \leq 1. \quad (1.2)$$

In general, the mechanical properties of the homogenized material depend on the full orientation tensor. However, if the local coordinate system is set to coincide with the principal fiber directions, the response of the material depends only on the two largest eigenvalues of the orientation tensor.

Based on this observation, ANSYS short fiber workflows build upon a three-step procedure:

1. Prior to and independently of the part simulation: Characterize the homogenized material for carefully selected sample orientations in the region of (λ_1, λ_2) defined by Equation 1.2 (p. 3). This produces a variable material whose properties depend on λ_1 and λ_2 .
2. For each given point on the injection molding mesh, decompose the fiber orientation tensor into its eigenvalues and eigenvectors.
3. Import the eigenvalues and eigenvectors of the orientation tensor as element orientations and material fields, respectively. Then map them onto the structural mesh. During the analysis, the element-wise material properties are obtained by querying the variable material for the value of (λ_1, λ_2) assigned to each element.

The next sections describe in more detail how to set up the workflow in each individual component, namely **Engineering Data**, **Material Designer**, **Injection Molding Data**, and **Mechanical**.

1.3. Constituents Material Data

The simulation of a short fiber reinforced composites requires the mechanical and thermal properties of the constituent materials, that is, the polymer and fibers. The structural analysis requires the properties of elasticity and possibly the secant coefficients of thermal expansion- in cases where thermal loads will be applied. For the thermal analysis, the thermal conductivity is required.

You will find the following popular materials provided as samples in the ANSYS Composite Materials library within **Engineering Data**:

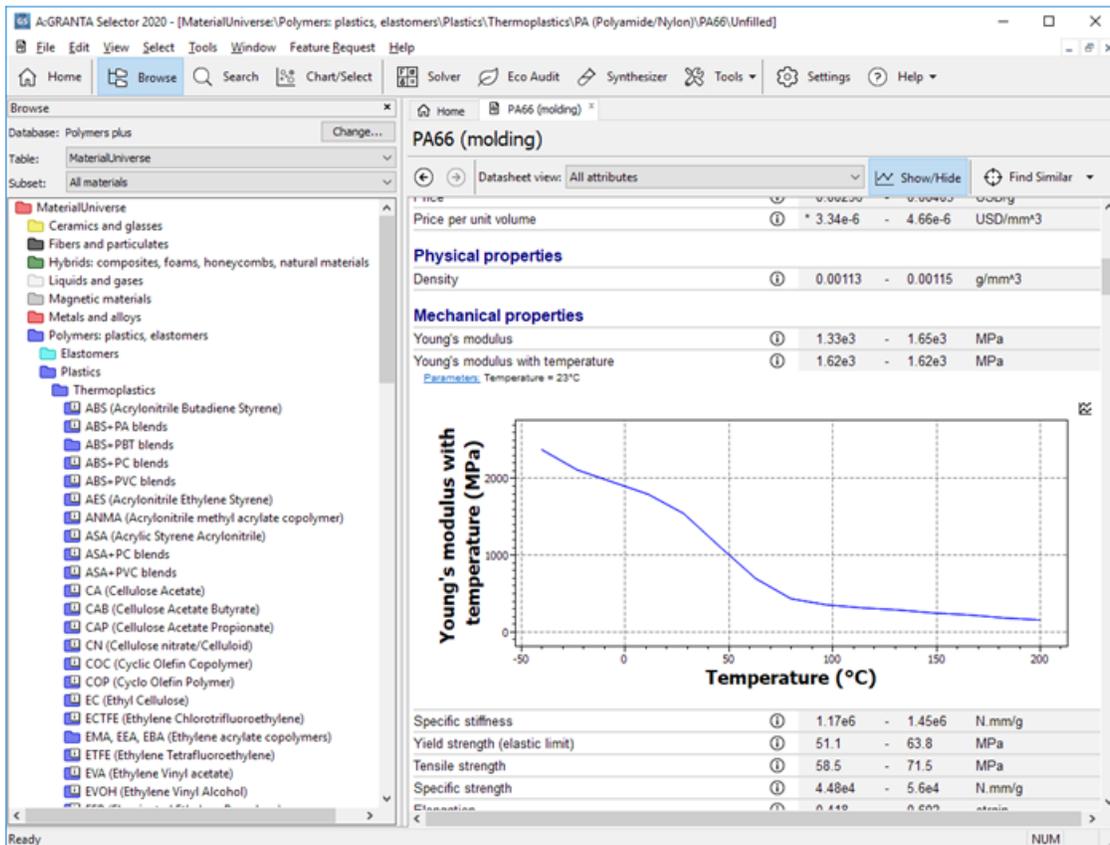
- Carbon Fiber (230 GPa)
- Carbon Fiber (290 GPa)
- Carbon Fiber (395 GPa)
- E-Glass
- Resin Epoxy
- Resin Polyester

The screenshot shows the ANSYS Engineering Data Sources window. The 'Engineering Data Sources' table lists various material categories. The 'Composite Materials' source (row 7) is selected, and its 'Outline of Composite Materials' is expanded to show a list of specific materials.

	A	B	C	
1	Data Source		Location	
5	Additive Manufacturing Materials			Additive
6	Geomechanical Materials			General
7	Composite Materials			Material
8	General Non-linear Materials			General
9	Explicit Materials			Material
10	Hyperelastic Materials			Material
11	Magnetic B-H Curves			B-H Cur
12	Thermal Materials			Material

	A	B	C	D
1	Contents of Composite Materials		Add	Source
2	Material			
3	Carbon Fiber (230 GPa)	+		Compos
4	Carbon Fiber (290 GPa)	+		Compos
5	Carbon Fiber (395 GPa)	+		Compos
6	E-Glass	+		Compos
7	Epoxy Carbon UD (230 GPa) Prepreg	+		Compos

You can access additional materials typically used through the **ANSYS GRANTA Selector** and import them into **Engineering Data**. For instance, the Material Universe library already offers a wide range of thermoplastics as well as several fiber types, such as glass, ceramic, metal, and others.



1.4. Microscale Homogenization in Material Designer

Material Designer offers two alternative approaches to computing homogenized properties of short fiber reinforced composites:

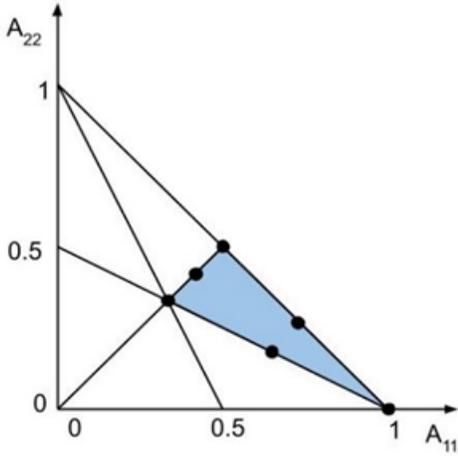
- An analytical (also called mean-field) approach, to rapidly compute the macroscopic properties of the composite. This approach is based on the Mori-Tanaka method combined with orientations averaging. (See *Analytical Homogenization* in the *Material Designer User's Guide*.)
- A finite element-based approach, more suited to perform an accurate and in-depth analysis of the composite. (See *Finite Element Based Homogenization* in the *Material Designer User's Guide*.)

In both cases, orthotropic elasticity, thermal expansion and thermal conductivity properties of the composite are computed based on its microstructure and constituents' properties. In particular, the following input quantities can be defined by the user:

- Isotropic thermo-elastic properties of the polymer (constant or temperature dependent)
- Isotropic or transversely isotropic thermo-elastic properties of the fibers (constant or temperature dependent)
- Fiber volume fraction, aspect ratio and orientation

You can also select temperature, fiber volume fraction, and the two largest eigenvalues of the orientation tensor as parameters for a variable analysis. The **Short Fiber Wizard** simplifies the process, guiding you to set up suitable sampling criteria of the parameter space. (See the section *Solve Variable Material* in the *Material Designer User's Guide*.)

Figure 1.1: Sampling of the orientation tensor space



You may find these related sections in the user's manual helpful as well:

- [Workbench Workflow Examples](#)
- [Model Type](#) and [Short Fiber Composite](#)

5. Optionally, specify the location of the Initial Stress File. Only the ANSYS IST element-based format is supported.
6. If needed, apply a **Rigid Transformation** to the input data by specifying the translation and/or rotation components.

10 Rigid Transformation			
11	Origin X	0	mm
12	Origin Y	0	mm
13	Origin Z	0	mm
14	Theta XY	0	degree
15	Theta YZ	0	degree
16	Theta ZX	0	degree

7. Connect to Mechanical:
 - To transfer fiber orientation tensor data to Mechanical, drag the **Setup** cell of the **Injection Molding Data** system and drop it onto the **Model** cell of a **Mechanical** system.
 - To transfer initial stress data to Mechanical, drag the **Setup** cell of the **Injection Molding Data** system and drop it onto the **Setup** cell of a **Mechanical System**. Alternatively, you can combine the two transfers by right-clicking the **Setup** cell of the **Injection Molding Data** cell and choosing **Transfer Data to New** in the context menu.

1.6. Setting Up the Mechanical System

A short fiber reinforced composite simulation involves most of the [general steps](#) found in any Mechanical analysis along with some additional steps and considerations described in the following sections:

- Import Element Orientations
- Import Material Fields
- Import Initial Stress
- Solving
- Review Results

The instructions below assume that a geometry is already available in the analysis system, whether imported from an external CAD system or created with SpaceClaim or DesignModeler. For additional information, refer to the [ANSYS Mechanical User's Guide](#).

1.6.1. Import Element Orientations

As a beta feature in 2020 R2, the **Imported Element Orientation** object enables you to import the principle fiber directions from the **Injection Molding Data System**, map them onto the structural mesh, and assign the elements coordinate system accordingly. Element orientations are imported as Euler angles expressed in degrees.

To this end, follow these steps in Mechanical:

1. Select the **Geometry** object in the tree and open the **Imported Element Orientation** folder.
2. Select the **Element Orientation** child object.

3. Scope the object to the geometry of interest.
4. As necessary, review and modify the remaining default settings of the properties in the **Details View**.
5. In the **Data View**, assign each of the imported Euler angles to the corresponding variable in Mechanical.

	Rotation XY-Plane (°)	Rotation YZ-Plane (°)	Rotation ZX-Plane (°)
1	Orientations:ThetaXY	Orientations:ThetaYZ	Orientations:ThetaZX

6. Right-click the **Imported Element Orientation** object and select the **Import Element Orientation** option from the context menu. This action maps the principle fiber directions to the elements of your model.

Note:

If your project involves complex fiber orientation distributions, the following settings are recommended due to current limitations in the software: Set the **Mapping Control** option to **Manual**, the **Weighting** option to **Distance Based Average**, and the **Number of Points** equal to 1.

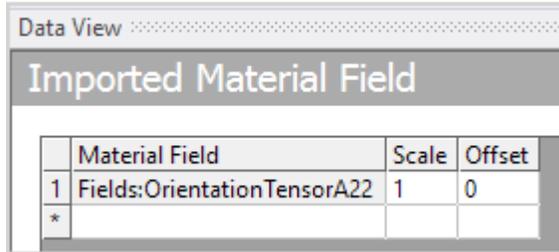
1.6.2. Import Material Fields

Using the **Imported Material Fields** interface, you can import the two largest eigenvalues of the fiber orientation tensor (from the **Injection Molding Data System**) and map them onto the structural mesh by doing the following:

1. Open the **Materials** folder as well as the **Imported Material Fields** object.
2. Select the **Imported Material Field** child object.
3. Scope the object to the geometry of interest.
4. As necessary, review and modify the remaining default settings of the properties in the **Details View**.
5. In the **Data View**, make sure to select the **OrientationTensorA11** identifier.

	Material Field	Scale	Offset
1	Fields:OrientationTensorA11	1	0
*			

- Right-click the **Imported Material Field** object and select the **Import Material Field** option from the context menu. This action maps the first eigenvalue of the orientation tensor to the elements of your model.
- Right-click the **Imported Material Field** object and select the **Duplicate** option from the context menu. In the **Data View**, make sure to select the **OrientationTensorA22** identifier.



Data View			
Imported Material Field			
	Material Field	Scale	Offset
1	Fields:OrientationTensorA22	1	0
*			

- Right-click the duplicated **Imported Material Field** object and select the **Import Material Field** option from the context menu. This action maps the second eigenvalue of the orientation tensor to the elements of your model.

1.6.2.1. Visualize the Homogenized Material Properties

The values of the homogenized material properties are calculated based on the values of the imported fiber orientation tensor eigenvalues. To inspect the spatial variation of the material properties, you can insert a **Material Plot** object from either the **Materials Context Tab** or by right-clicking the **Materials** object and selecting **Insert > Material Plot**.

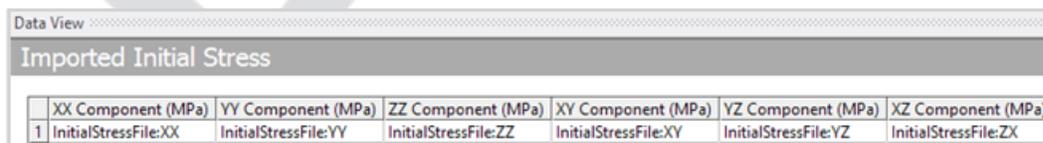
You will find further details in the [Mechanical User's Guide](#).

1.6.3. Import Initial Stress

When an **Initial Stress File** is specified in the **Injection Molding Data** setup component, you can insert an **Imported Initial Stress** object to represent the transfer. Note that when connecting the **Injection Molding Data** component to the **Setup Cell** of a **Mechanical System**, an **Imported Load** folder appears under the **Environment** folder by default.

Follow these steps in Mechanical:

- To add an imported load, click the **Imported Load** folder to make the **Environment** context tab available. Or right-click the **Imported Load** folder and select the **Initial Stress** from the context menu.
- Scope the object to the geometry of interest.
- In the **Data View**, assign each of the imported initial stress components to the corresponding variable in Mechanical.



Data View						
Imported Initial Stress						
	XX Component (MPa)	YY Component (MPa)	ZZ Component (MPa)	XY Component (MPa)	YZ Component (MPa)	XZ Component (MPa)
1	InitialStressFile:XX	InitialStressFile:YY	InitialStressFile:ZZ	InitialStressFile:XY	InitialStressFile:YZ	InitialStressFile:ZX

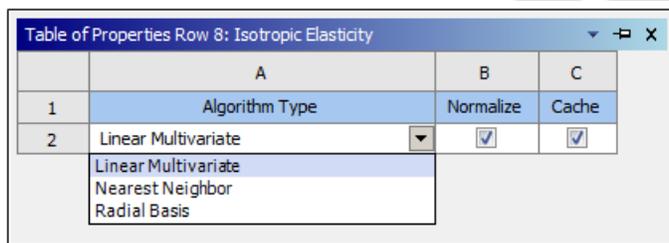
4. As necessary, review and modify the remaining default settings of the properties in the **Details View**. You can view contour plots of XX/YY/ZZ/XY/YZ/ZX components by changing the **Component** option in the **Graphics Control** group. (It defaults to a **Vector Principal** plot).
5. Right-click the **Imported Load** object and select the **Import Load** option from the context menu.

You will find further details in the [Mechanical User's Guide](#).

1.6.4. Solving

In some circumstances, the solver might fail to evaluate the thermal and/or elastic properties of the homogenized material at some query points. Specifically, the solver fails when extrapolation occurs. (You will find more information in the [Material Reference](#) section of the *Mechanical APDL User's Guide*.)

In such cases, as a workaround, you can select an interpolation algorithm different from Linear Multivariate in **Engineering Data**. For additional information, see the [Material Data](#) section of the *Workbench User's Guide*.



	A	B	C
1	Algorithm Type	Normalize	Cache
2	Linear Multivariate	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

1.6.5. Review Results

The analysis type determines which results are available after the solution is completed. The section [Using Results](#) in the *Mechanical User's Guide* lists the various results available for postprocessing. The following result types are particularly relevant to post-processing short fiber reinforced composites simulations.

Elemental Triads Plot

With the **Coordinate Systems** plots, you can display elemental result rotations as Euler rotated triads at each element's centroid, or you can display them as contours that represent a Euler rotation angle about an individual elemental axis. This type of plot is particularly useful for visualizing the mapped principal fiber orientations.

Stress along the first principal fiber direction

In a structural analysis, you can display the stress along the first principal fiber direction by following these steps:

1. Insert a **Normal Stress** result object.
2. Set the **Orientation** option in the **Details View** to X Axis.
3. Set the **Coordinate System** option in the **Details View** to **Solution Coordinate System**.

Stress in the transverse fiber direction

In a structural analysis, you can display the absolute shear stress on the plane transverse to the first principal fiber direction by following these steps:

1. Insert a **User Defined Result** object.
2. Define the **Expression** as " $\sqrt{SXY^2+SXZ^2}$ ".
3. Set the **Coordinate System** option in the **Details** view to **Solution Coordinate System**.