Autodesk® Moldflow® Insight 2012

AMI Fiber Orientation Analysis Results

Autodesk®

Revision 1, 23 March 2012.

This document contains Autodesk and third-party software license agreements/notices and/or additional terms and conditions for licensed third-party software components included within the product. These notices and/or additional terms and conditions are made a part of and incorporated by reference into the Autodesk Software License Agreement and/or the About included as part of the Help function within the software.

Contents

Fiber Orientation analysis results

1

This help topic specifies the results for a Fill analysis or Fill+Pack analysis sequence on a thermoplastic material with fiber or filler content.

Text based results

The following table lists the text results generated by using the fiber option in a Fill analysis or Fill+Pack analysis sequence.

Results
Analysis Log
Results Summary
Analysis Check
Machine Setup

Graphical results

The following table lists the graphical text results generated by using a material with fiber or filler content in a Fill analysis or Fill+Pack analysis sequence, and indicates whether each result is supported for the following analysis technologies:

-  Midplane
-  Dual Domain
-  3D

For more information about a result, including how to interpret the display, click on the result name.

Result	Analysis technology
Average fiber orientation result on page 3	
Fiber orientation tensor result on page 5	
Linear thermal expansion coefficient results on page 7	
Poisson's ratio (fiber) result on page 8	
Poisson's ratio (v12) result on page 10	

Result	Analysis technology
Poisson's ratio (v13) (3D fiber) result on page 12	
Poisson's ratio (v23) (3D fiber) result on page 13	
Shear modulus (fiber) result on page 14	
Shear modulus (G12) result on page 16	
Shear modulus (G13) (3D fiber) result on page 18	
Shear modulus (G23) (3D fiber) result on page 20	
Tensile modulus, principal direction results on page 22	
Tensile modulus, principal direction (fiber)	

NOTE: Fiber results from a Fill analysis or a Fill+Pack analysis sequence are created per the material selected. For example, if you select a material with fiber or filler content, Fiber results will be generated.

Average fiber orientation result

2

The Average fiber orientation result is generated by a Fill or Fill+Pack analysis using Midplane or Dual Domain analysis technology, when the option to perform **Fiber orientation analysis if fiber material** is enabled.

It shows the orientation of fibers during the injection molding process, averaged over the thickness.

Using this result

The Fiber orientation analysis calculates the layer-based fiber orientation tensor at each time-step throughout the duration of the analysis. This tensor is expressed in six components in the local coordinate system of each element.

The average fiber orientation tensor is a tensor whose components are the average of the components of the layer-based fiber orientation tensor over thickness.

The average fiber orientation is expressed in the global coordinate system.

The Average fiber orientation result is output as a time series result, so that you can see the movement of fiber orientation throughout the injection molding process. The default plot shows tensor first principal values over the analysis duration.

The diagonal components of a fiber orientation tensor represent the strength of alignment in the respective directions. The values of diagonal components range between 0 and 1, and the sum of all three diagonal components is 1. The off-diagonal components of a fiber orientation tensor represent the amount that alignments vary from the coordinate axes, and they are zero when coordinate axes coincide with the principal directions of the orientation tensor.

The principal directions are numbered in the order of the respective principal values, from the largest to the smallest. The first principal direction represents the direction along which the most fibers are aligned, and the third principal direction represents the one along which the fewest fiber are aligned. The larger the principal value is, the stronger the alignment in the corresponding principal direction is.

TIP: To more easily interpret fiber orientation results, it is recommended to check principal values or tensor components as well as principal directions. Right-click the result name in the **Study Tasks** pane and select **Properties**:

- Select the **Methods** tab to change the result display method. The default **Tensor as axes** display method and other tensor display method options plot both value and principal direction results. The **Shaded** and **Contour** display methods plot only value results.

- Select the **Tensor** tab to change the tensor principal value (direction) and tensor component displayed on the model.

Things to look for

- To view the history of specific tensor components at specific locations on the model, create a new XY plot of the Average fiber orientation result.
- Inconsistent orientation of fibers.

Fiber orientation tensor result

3

The Fiber orientation tensor result shows the orientation tensor (degree of orientation) at the end of the injection molding process.

This result is generated by a Fiber orientation analysis.

Using this result

Using Midplane or Dual Domain analysis technology, the Fiber orientation analysis calculates the layer-based fiber orientation tensor at each time-step throughout the Fiber orientation analysis. Using 3D analysis technology, the Fiber orientation analysis creates nodal and elemental fiber orientation tensor results. This tensor is symmetric, thus it is expressed with six components in each laminate of each shell element of a Midplane or Dual Domain mesh model, or at each node and on each tetrahedral element in a 3D mesh model.

The fiber orientation tensor output is the fiber orientation tensor result at the end of the injection molding process.

The Fiber orientation tensor result shows the probability of fiber alignment in the specified principal direction. A high probability of fiber alignment in the specified principal direction will be indicated by a value of close to 1 on the result scale, whereas a low probability is indicated by a value close to 0. The fiber orientation tensor in the first principal direction is the most useful result to view.

NOTE: The first principal direction is close to the material flow direction in most cases, but it may not always coincide with the flow direction.

The fiber orientation tensor is expressed in the global coordinate system, and is also used to calculate the mechanical/thermal property as well as the residual stress in the cavity.

The diagonal components of a fiber orientation tensor represent the strength of alignment in the respective directions. The values of diagonal components range between 0 and 1, and the sum of all three diagonal components is 1. The off-diagonal components of a fiber orientation tensor represent the amount that alignments vary from the coordinate axes, and they are zero when coordinate axes coincide with the principal directions of the orientation tensor.

The principal directions are numbered in the order of the respective principal values, from the largest to the smallest. The first principal direction represents the direction along which the most fibers are aligned, and the third principal direction represents the one along which the fewest fiber are aligned. The larger the principle value is, the stronger the alignment in the corresponding principal direction is.

TIP: To more easily interpret fiber orientation results, it is recommended to check principal values or tensor components as well as principal directions. Right-click the result name in the **Study Tasks** pane and select **Properties**:

- Select the **Methods** tab to change the result display method. The default **Tensor as axes** display method and other tensor display method options plot both value and principal direction results. The **Shaded** and **Contour** display methods plot only value results.
- Select the **Tensor** tab to change the tensor principal value (direction) and tensor component displayed on the model.

ATTENTION: Using 3D analysis technology, the default Fiber orientation tensor result displayed is based on nodal data. Using nodal data improves the display of the fiber orientation profile through the thickness of the part. This is most easily seen when the Fiber orientation tensor result is displayed using the plot type **Probe XY plot**. This result based on elemental data is still created, but it is not displayed by default. To display it, click **Results tab > Plots panel > New Plot** and select Fiber orientation tensor on elements from the list of available results.

Things to look for

- The fiber orientation tensor result provides the final results of fiber orientation, at the end of the analysis.
- Inconsistent alignment/orientation of fibers. There should be a high probability of fiber alignment in the first principal direction.

Linear thermal expansion coefficient results 4

The Linear thermal expansion coefficient result indicate any shrinkage that has occurred in relation to temperature changes.

A Pack analysis produces the following results:

- Linear thermal expansion coefficient in first direction
- Linear thermal expansion coefficient in second direction
- Linear thermal expansion coefficient in third direction

Fiber orientation analysis produces the following results using Midplane and Dual Domain analysis technology:

- Linear thermal expansion coefficient in first direction (fiber)
- Linear thermal expansion coefficient in second direction (fiber)

TIP: Click **Result > New Plot** and create this result as an XY or Path plot also so that you can view the linear thermal expansion coefficient at a particular element.

Using these results

The material should expand less in the direction of flow (first principal direction) than perpendicular to the fiber orientation (second principal direction). If the molecules are aligned in the first and second principal directions, then the linear thermal expansion coefficient will be different in each principal direction. If the molecules are randomly aligned, then there will be a uniform linear thermal expansion coefficient in each principal direction.

Using Midplane or Dual Domain analysis technology, the linear thermal expansion coefficient is recorded for each laminate in the model over the duration of the analysis. Therefore, each laminate in the model will have different thermal expansion results. You can check the expansion coefficient in each laminate by animating the default contour plot, which will animate the result over the Normalized Thickness.

These results are very important when proceeding to perform a Stress analysis as the results will be used in the thermal loading calculation.

TIP: You can view the contour plot at a single laminate on the model. To do so, click **Result > Plot Properties**. The **Plot properties** dialog opens. Select the **Animation** tab, and from the **Animate result over** drop-down list, select **Single dataset**.

Poisson's ratio (fiber) result

5

The Poisson's ratio (fiber) result is a fiber-over-thickness (FOT) averaged value, therefore, it is an element-based value averaged over all the laminates of each element.

This result is generated from a Fiber orientation Pack analysis.

Using this result

The Poisson's Ratio (fiber) result indicates the average strain in the second principal direction caused by the stress in the first principal direction.

The Poisson's ratio, (v12) result is a laminate-based value, therefore, you can view the Poisson's ratio distribution for each laminate in the model. You can check this result for more detailed information on the Poisson's ratio distribution.

Poisson's ratios are mechanical property values. The distribution of this mechanical property is used by the structural analysis for its performance evaluation in a Stress analysis.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients. In models analysed using Midplane or Dual Domain analysis technologies, because of the plain stress assumption in the shell structure analysis in Warp, only 4 mechanical constants (tensile modulus in first/second principal directions, Poisson ratio v12, shear modulus G12) are necessary.

The **Orthotropic set** option selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once.

NOTE: To access the **Orthotropic set** option ensure you have selected an analysis sequence that includes Fill+Pack.

1 Click **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2 If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.
3 Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens

4 Click **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.

Poisson's ratio (v12) result

6

The Poisson's ratio (v12) result indicates the strain in the second principal direction caused by the stress in the first principal direction.

In general, vij = Poisson's ratio for transverse strain in the j-direction when stressed in the i-direction.

This result is generated from a Fiber orientation Pack analysis.

TIP: Click **Results tab > Plots panel > New Plot** and create this result as an XY or Path plot also so that you can view the Poisson's ratio distribution at a particular element, or with respect to the model geometry.

Using this result

For a Midplane and Dual Domain analysis, the Poisson's ratio (v12) result is recorded for each laminate in the model over the duration of the analysis. Therefore, each laminate in the model will have different results. You can check the Poisson's ratio in each laminate by animating the default contour plot, which will animate the result over the Normalized thickness.

TIP: You can view the contour plot at a single laminate on the model. To do so, click **Results tab > Properties panel > Plot Properties**. The **Plot properties** dialog opens. Select the **Animation** tab, and from the **Animate result over** drop-down list, select **Single dataset**.

This result is very important when proceeding to perform a Stress analysis because the results will be used in the thermal loading calculation.

For a 3D Fiber orientation analysis, the Poisson's ratio (v12) result is recorded for each tetrahedral element in the model at the end of the analysis.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients. In models analysed using Midplane or Dual Domain analysis technologies, because of the plain stress assumption in the shell structure analysis in Warp, only 4 mechanical constants (tensile modulus in first/second principal directions, Poisson ratio v12, shear modulus G12) are necessary.

The **Orthotropic set** option selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once. In a Fiber analysis, the complete set of thermo-mechanical properties with orthotropic assumption is necessary for a Warp analysis using 3D analysis technology. These properties are element-based, so each tetrahedral or beam element has its own orthotropic set of properties.

NOTE: To access the **Orthotropic set** option ensure you have selected an analysis sequence that includes Fill+Pack.

1 Click **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2 If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.
3 Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens
4 Click **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.

Poisson's ratio (v13) (3D fiber) result

7

The Poisson's ratio (v13) result is generated from a Fiber orientation Pack analysis using 3D analysis technology.

Using this result

The Poisson's ratio (v13) result indicates the strain in the third principal direction caused by the stress in the first principal direction. In general, vij = Poisson's ratio for transverse strain in the j-direction when stressed in the i-direction.

The Poisson's ratio (v13) result is recorded for each tetrahedral element in the model at the end of the analysis.

TIP: Click **Results tab > Plots panel > New Plot** and under **Plot type** click **Path plot** so that you can view the Poisson's ratio distribution with respect to the model geometry.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients.

The Orthotropic set option selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once. In a Fiber analysis, the complete set of thermo-mechanical properties with orthotropic assumption is necessary for a Warp analysis using 3D analysis technology. These properties are element-based, so each tetrahedral or beam element has its own orthotropic set of properties.

NOTE: To access the **Orthotropic set** option, ensure that you have selected an analysis sequence that includes Fill+Pack.

1. Click **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2. If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.
3. Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens
4. Click **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.

Poisson's ratio (v23) (3D fiber) result

8

The Poisson's ratio (v23) result indicates the strain in the third principal direction caused by the stress in the second principal direction.

In general, vij = Poisson's ratio for transverse strain in the j-direction when stressed in the i-direction.

This result is generated from a Fiber orientation Pack analysis using 3D analysis technology.

Using this result

The Poisson's ratio (v23) result is recorded for each tetrahedral element in the model at the end of the analysis.

TIP: Click **Results tab > Plots panel > New Plot** and under **Plot type**, click **Path plot** so that you can view the Poisson's ratio distribution with respect to the model geometry.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients.

The **Orthotropic set** option selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once. In a Fiber analysis using 3D analysis technology, the complete set of thermo-mechanical properties with orthotropic assumption is necessary for a Warp analysis using 3D analysis technology. These properties are element-based, so each tetrahedral or beam element has its own orthotropic set of properties.

NOTE: To access the **Orthotropic set** option ensure you have selected an analysis sequence that includes Fill+Pack.

1. Click **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2. If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.
3. Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens.
4. Cick **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.

Shear modulus (fiber) result

9

The Shear modulus, sometimes called the rigidity modulus, refers to the change produced by a tangential stress, and provides a measure of how “stiff” the material is.

The Shear modulus (fiber) result is generated from a Fiber orientation Pack analysis using Midplane or Dual Domain analysis technology.

Using this result

The Shear modulus (fiber) result is a fiber-over-thickness (FOT) averaged value, therefore, it is an element-based value averaged over all the laminates of each element.

The Shear modulus (G12) result is a laminate-based value, therefore, you can view the shear modulus distribution for each laminate in the model. You can check this result for more detailed information on the shear modulus distribution.

Shear modulus is a mechanical property value. The distribution of this mechanical property is used by the structural analysis for its performance evaluation in a Stress analysis.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients. In models analysed using Midplane or Dual Domain analysis technology, because of the plain stress assumption in the shell structure analysis in Warp, only 4 mechanical constants (tensile modulus in first/second principal directions, Poisson ratio v12, shear modulus G12) are necessary.

The **Orthotropic set** option selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once.

NOTE: To access the **Orthotropic set** option ensure you have selected an analysis sequence that includes Fill+Pack.

1 Click **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2 If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.

3 Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens
4 Click **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.

Shear modulus (G12) result 10

The Shear modulus (G12) result shows the shear strain applied in the XY direction, coinciding with the fiber orientation direction.

The Shear modulus, sometimes called the rigidity modulus, refers to the change produced by a tangential stress, and provides a measure of how “stiff” the material is.

This result is generated from a Fiber orientation Pack analysis.

TIP: Click **Results tab > Plots panel > New Plot** and under **Plot type**, click **XY plot** or **Path plot**, so that you can view the shear modulus at a particular element.

Using this result

For a Midplane and Dual Domain analysis, the Shear modulus (G12) result is recorded for each laminate in the model over the duration of the analysis. Therefore, each laminate in the model will have different results. You can check the shear modulus in each laminate by animating the default contour plot, which will animate the result over the Normalized thickness.

TIP: You can view the contour plot at a single laminate on the model. To do so, click **Results tab > Properties panel > Plot Properties**. The **Plot properties** dialog opens. Select the **Animation** tab, and from the **Animate result over** drop-down list, select **Single dataset**.

This result is very important when proceeding to perform a Stress analysis as the results will be used in the thermal loading calculation.

For a 3D Fiber orientation analysis, the Shear modulus (G12) result is recorded for each tetrahedral element at the end of the analysis.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients. In models analysed with Midplane or Dual Domain analysis technologies, because of the plain stress assumption in the shell structure analysis in Warp, only 4 mechanical constants (Tensile modulus in first/second principal directions, Poisson ratio v12, Shear modulus G12) are necessary.

The **Orthotropic set** option selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once. In a Fiber analysis using 3D analysis technology, the complete set of thermo-mechanical properties with orthotropic assumption is necessary for a Warp analysis using 3D analysis technology. These properties are element-based, so each tetrahedral or beam element has its own orthotropic set of properties.

NOTE: To access the **Orthotropic set** option ensure you have selected an analysis sequence that includes Fill+Pack.

1. Click **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2. If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.
3. Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens
4. Click **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.

Shear modulus (G13) (3D fiber) result 11

The shear modulus, sometimes called the rigidity modulus, refers to the change produced by a tangential stress, and provides a measure of how “stiff” the material is.

The Shear modulus (G13) result indicates the shear strain applied in the XZ direction, or the direction perpendicular to the fiber orientation direction.

This result is generated from a Fiber orientation Pack analysis using 3D analysis technology.

Using this result

The Shear modulus (G13) result is recorded for each tetrahedral element in the model at the end of the analysis.

TIP: Click **Results tab > Plots panel > New Plot** and under **Plot type**, click **Path plot** so that you can view the shear modulus distribution with respect to the model geometry.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients.

The **Orthotropic set** option (found in the **Fiber Orientation Solver Parameters** dialog under **Composite property calculation options > Fiber-filled property output**) selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once. In a Fiber analysis, the complete set of thermo-mechanical properties with orthotropic assumption is necessary for a Warp analysis using 3D analysis technology. These properties are element-based, so each tetrahedral or beam element has its own orthotropic set of properties.

NOTE: To access the **Orthotropic set** option ensure you have selected an analysis sequence that includes Fill+Pack.

1 Click **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2 If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.

3 Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens
4 Click **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.

Shear modulus (G23) (3D fiber) result 12

The Shear modulus (G23) result indicates the shear strain applied in the YZ direction (second and third principal directions).

The shear modulus, sometimes called the rigidity modulus, refers to the change produced by a tangential stress, and provides a measure of how “stiff” the material is.

This result is generated from a Fiber orientation Pack analysis using 3D analysis technology.

Using this result

The Shear modulus (G23) result is recorded for each tetrahedral element in the model at the end of the analysis.

TIP: Click **Results tab > Plots panel > New Plot** and under **Plot type**, click **Path plot** so that you can view the shear modulus distribution with respect to the model geometry.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients.

The **Orthotropic set** option selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once. In a Fiber analysis, the complete set of thermo-mechanical properties with orthotropic assumption is necessary for a Warp analysis using 3D analysis technology. These properties are element-based, so each tetrahedral or beam element has its own orthotropic set of properties.

NOTE: To access the **Orthotropic set** option ensure you have selected an analysis sequence that includes Fill+Pack.

1 Click **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2 If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.
3 Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens

4 Click **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.

Tensile modulus, principal direction results 13

The Tensile modulus, principle direction results indicate how much stress is needed to cause a unit of movement.

This help topic covers the following Fiber orientation Pack analysis results:

- Tensile modulus in first principal direction
- Tensile modulus in second principal direction

Using these results

The Tensile modulus in first and second principal direction results are recorded for each laminate in the model over the duration of the analysis. Therefore, each laminate in the model will have different results. You can check the tensile modulus in each laminate by animating the default contour plot, which will animate the result over the Normalized thickness.

The first principal direction coincides with the fiber orientation first principal direction, and is determined by the Fiber orientation analysis. The second principal direction is perpendicular to the first principal direction.

TIP: You can view the contour plot at a single laminate on the model. To do so, click **Results tab > Properties panel > Plot Properties**. The **Plot properties** dialog opens. Select the **Animation** tab, and from the **Animate result over** drop-down list, select **Single dataset**.

TIP: Click **Results tab > Plots panel > New Plot** and under **Plot type**, click **XY plot** or **Path plot**, so that you can view the tensile modulus in the first or second principal direction at a particular element.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients. In Midplane/Dual Domain models, because of the plain stress assumption in the shell structure analysis in warpage, only 4 mechanical constants (Tensile modulus in first/second principal directions, Poisson ratio v12, Shear modulus G12) are necessary.

The **Orthotropic set** option selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once.

NOTE: To access the **Orthotropic set** option ensure you have selected an analysis sequence that includes Fill+Pack.

1 Click  **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2 If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.
3 Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens
4 Click **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.
