Autodesk® Moldflow® Insight 2012

AMI Warp 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

Warp analysis results

1

This topic specifies the results for a Warp analysis.

Text based results

The following table lists the text results that are created for each Warp analysis.

Results
Warp analysis log on page 27
Summary
Analysis Check

Graphical results

The following table lists the graphical results that are created for a Warp analysis and indicates whether each result is supported by the following analysis technologies:

- Midplane
- Dual Domain
- 3D

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

Result	Available for analysis technology
Deflection result on page 11 [1]	
Deflection result on page 11 [1]	
Deflection result on page 11 [1]	
Deflection result on page 11 [1]	
Deflection result on page 11 [1]	
Deflection result on page 11 [1]	

[1] All deflection results are available as net deflection and X, Y and Z component deflection plots.

Result	Available for analysis technology
Mode shape result on page 18	
Mode shape result on page 18 [2]	
Stress tensor (warp) result on page 26 [2]	
Strain in first principal direction result on page 23 [2]	
Strain in second principal direction result on page 24 [2]	
Strain tensor (warp) result on page 25 [2]	
Change in refractive index after warpage result on page 9 [3]	
Phase Shift for light coming from +/- Z direction result on page 20 [3]	
Retardation for light coming from +/- Z direction result on page 22 [3]	
Maximum shear stress result on page 16	
Anisotropic shrinkage result on page 7	
Isotropic shrinkage result on page 14	
Parallel shrinkage, before warpage result on page 19	
Perpendicular shrinkage, before warpage result on page 21	
Bending curvature result on page 8	
Material orientation results on page 15	

[2] Only available if **Stress result(s) to output** setting in the Process Settings Wizard has been changed from the default setting of **None**.

[3] Only available if using a material with optical properties.

Warpage Visualization Tools 2

Specify a warpage plane of reference from which warpage part deflection results can be viewed.

Click a **Deflection** (Warp) result to activate the Warpage Visualization Tools. Click three locations on the model to define the anchor plane. The warpage result is automatically updated with the defined plane of reference.

NOTE: You must have already run a Warp analysis to use the Warpage Visualization Tools.

TIP: Right-click the Warp result name in the **Study Tasks** pane and select **Properties > Deflection**, and then change the warpage Color and Scale factor of the displayed result.

Anchor planes

After a Warp analysis has been run, a reference plane against which warpage will be measured must be defined. This is the anchor plane, which is defined by three points called anchor points being selected on an undistorted part.

NOTE: The anchor plane is set automatically by the warpage solver during an analysis. To include a non-default anchor plane in an Autodesk Moldflow Results file (*.mfr), you must select a new anchor plane before exporting results.

Where to locate the anchor plane

Consider the position of the anchor plane carefully when viewing warpage results. Locate the plane across a flat part section where you can easily visualize the deflections, and where the implications of the deflections can be most clearly interpreted.

The possible locations for an anchor plane include the following:

- The base of a part that is required to lie flat on a surface.
- The joining plane to a mating component.
- The feet of the part.

NOTE: The anchor plane does not have to be parallel to the XY plane.

Choosing the three defining nodes

The anchor plane is defined by selecting three locations (nodes) on the part. Select the three nodes over a large area to define an easily visualized reference plane. This plane is used to measure the deflections, as shown below.

The sequence in which the anchor points are defined is important as this affects the warpage results.

NOTE: The anchor points do not have to be on the corners of the part. Typically they are placed at fixing points of the finished assembly, or along edges where two parts meet.

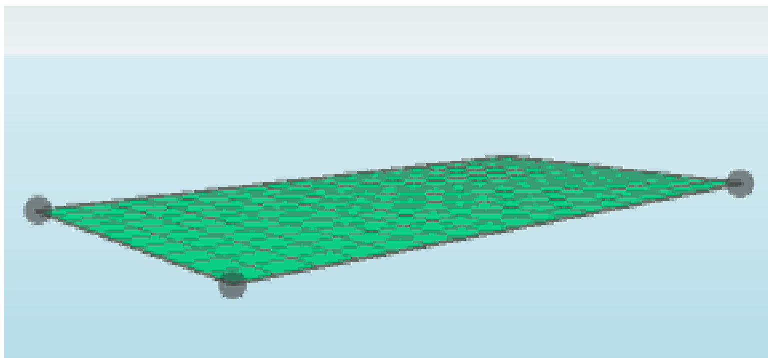

Figure 1: Three anchor points set on an undistorted part. The distorted part must be oriented in relation to this plane for measurement.

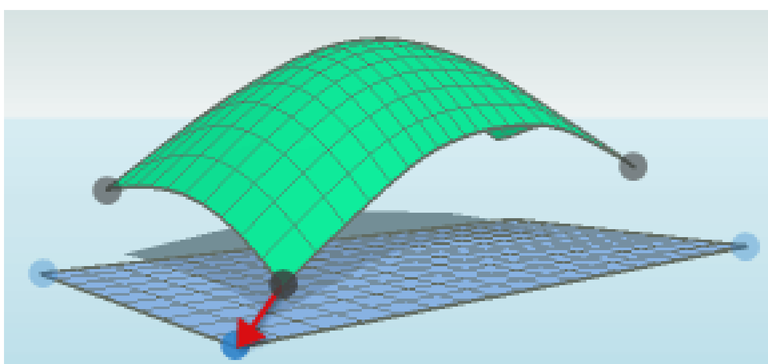

Figure 2: The first anchor point defined is the front center point. The corresponding point on the part to be measured must always align with this anchor point.

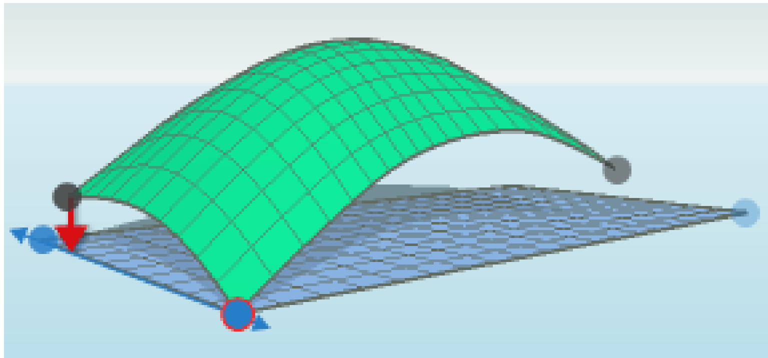

Figure 3: The second anchor point defines a line from the first anchor point. The corresponding second point on the part to be measured must be placed along this line while still maintaining the positioning of the first anchor point.

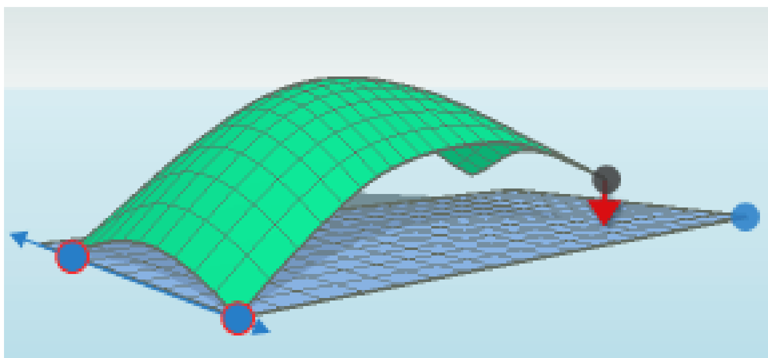

Figure 4: The third anchor point defines the anchor plane. The third point on the part to be measured is now placed on this plane while maintaining the position of the first two anchor points.

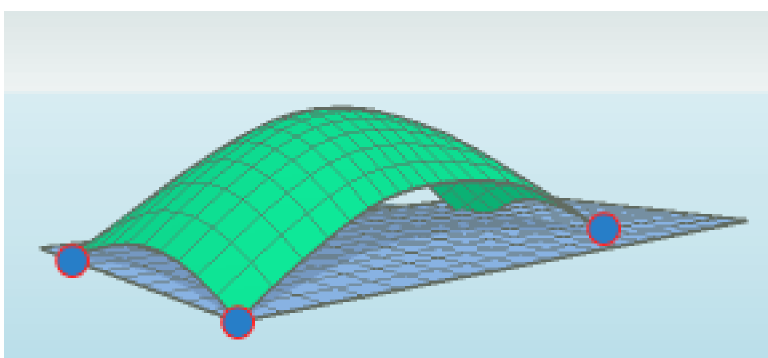

Figure 5: The part is now oriented correctly in relation to the anchor plane.

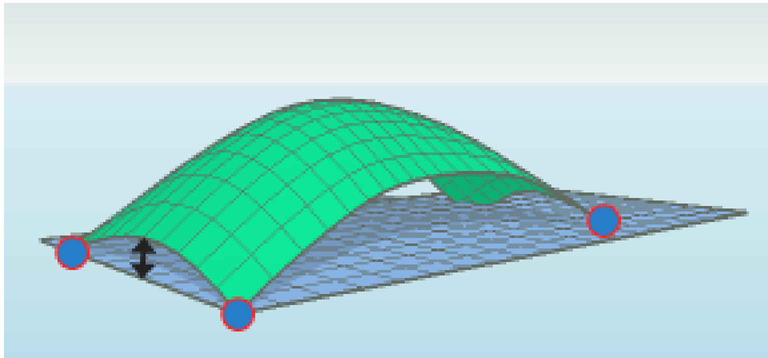

Figure 6: The distance from the anchor plane to the part is now a repeatable measurement of the warpage

Adjusting the anchor plane definition

The three locations you have selected to define the anchor plane are shown by three corresponding symbols. You can redefine the anchor plane by dragging one or more of the symbols to a new location on the part. As soon as you release the left mouse button to place the symbol at the new location, the deflection results will automatically be updated.

Anisotropic shrinkage result

4

The Anisotropic shrinkage result shows half the difference between the perpendicular and parallel shrinkage value for each element.

The formula to calculate this result is $(SH_{perpendicular} - SH_{parallel})/2$.

NOTE: Parallel shrinkage is defined to be in the direction of greatest fiber orientation for fiber-filled materials and in the flow direction of the material for non-fiber-filled materials. Perpendicular shrinkage is defined to be the inplane direction perpendicular to the direction of parallel shrinkage.

When using unfilled or particle filled materials, the anisotropic shrinkage may be positive (indicating that the perpendicular shrinkage is dominant), or negative (indicating the parallel shrinkage is dominant). Fiber-filled materials almost always have positive anisotropic shrinkage .

This result uses shrinkage computed at each element before warpage is calculated, therefore these shrinkage values represent the tendency to shrink at each element before the structural influence of the surrounding part is considered.

Using this result

The difference between the parallel and perpendicular shrinkage in each element is an indication of the warpage due to orientation. The greater the difference, the greater the warpage is likely to be.

The distribution of the anisotropic shrinkages is the most useful result to examine when considering the warpage problems due to the orientation effect.

You may want to consider ways to reduce warpage due to differential shrinkage in your part.

Bending curvature result

5

The Bending Curvature result shows the predicted amount of curve on the part.

A large bending curvature value indicates a large bend in the part.

The Bending Curvature result is given by the difference between the top and bottom shrinkage of an element divided by the thickness of the element:

Abs⊠TOP–SBOTTOMthickness

Using this result

As can be seen from the definition of the result shown above, the bending curvature result increases when the difference between the top and bottom shrinkages increases, and when the part thickness decreases. The largest bending curvature values therefore occur in thin sections of the part, with large differences in shrinkage between the top and bottom of the part.

The Bending Curvature result is therefore a useful result to examine when considering warpage problems due to differential cooling. Those areas of the model with larger bending curvature values are more prone to out-of-plane deflections resulting from differential cooling at the top and bottom of the element.

Change in refractive index after warpage result

6

The Change in refractive index after warpage result displays the difference between the refractive index after warpage and the material's nominal refractive index as it appears in the material database.

This difference is a result of stresses within the part.

This result is a tensor value. You can change the way it is displayed by modifying the plot properties. To best view this result, select **Results tab > Properties panel > Plot Properties**, select the **Methods** tab, and select **Tensor as axes** or **Tensor as ellipsoids** in the **Selection** box.

The Change in refractive index after warpage result is created at the end of a 3D Warp analysis.

NOTE: This result is generated only if you have selected the **Birefringence analysis if material data includes optical properties** checkbox in the Fill+Pack page of the Process Settings wizard and the material has measured optical properties.

Using this result

This result can be used qualitatively to detect changes in refractive index that are caused by stresses in the part after deformation. These stresses originate from the in-mold residual stresses that were not able to relax after ejection, or from stresses caused by the part cooling and deforming. A high value in this result may indicate that you need to move the injection location, alter processing conditions to reduce the stresses in the part (such as increasing the cooling time), or modify the geometry of the model so that stresses are concentrated to optically unimportant sections of the part.

NOTE: The purpose of the part determines what is an acceptable change in refractive index. A value of 0.1 may be acceptable for general purposes, but unacceptably large for optical purposes.

Things to look for

Birefringence occurs when the change in refractive index tensor has different values in different principal directions (i.e., the result is displayed as ellipsoids). Birefringence has two visible effects, depending on the orientation of the tensor. Depending on the purpose of the part, one effect may be more important.

Double images If the change in refractive index tensor has no axis along the direction of the incident light, then unpolarized light may appear as a double image when looking through the part.

Polarization effects If one of the axes of the change in refractive index tensor is parallel to the incident light, no double image will appear, but instead the polarization of the light will change. The effect is proportional to the difference in the lengths of the two orthogonal components of the tensor. A change in polarization may be noticeable as colored bands like those that may be seen on the surface of a bubble.

If the change in refractive index tensors are symmetrical in each axis (i.e., the tensor is displayed as spheres) then birefringence cannot occur, though a lens effect is still possible if the magnitude of the tensor varies across the part.

7 Deflection result

The deflection result shows the deflection at each node of the part (Warp or Stress analysis), or each node of the wire or paddle (Microchip Encapsulation analysis).

It is based on a *best fit* technique, where the original geometry and the deformed geometry are overlayed in such a way that they best fit together, or based on a defined anchor plane. This can be defined using **Results tab > Warpage panel > Visualize**.

There are a number of possible variants of the deflection result according to paddle (Microchip Encapsulation analysis), based on a *best fit*:

Analysis type
The result name may indicate the type of analysis that was run, that is, either small deflection or large deflection. If this is not indicated in the result name, then the results will apply to a small deflection analysis.

Net vs component deflections
You can view the net deflections at each node, or the component of the deflection along either the X, Y or Z axis. The axis directions are determined by the defined anchor plane and are indicated in the anchor plane symbols.

All effects vs warpage contributors
There are four sets of deflection results. To create these results, run a small deflection Warp analysis and select the **Isolate cause of warpage** option on the Warp Settings page of the Process Settings Wizard.

- **All effects** — The total deflection at each node.
- **Differential shrinkage** — The component of the total deflection (warpage) at each node attributable to differential shrinkage.
- **Orientation effects** — The component of the total deflection (warpage) at each node attributable to orientation.
- **Differential cooling** — The component of the total deflection (warpage) at each node attributable to differential cooling.
- **Corner effects** — The component of the total deflection (warpage) at each node attributable to corner effects.

Using this result

The net deflection plot shows you the total predicted deformation of the part, based on the default *best fit* technique, as explained above, or the defined anchor plane.

If the deflection values are quite small, you can magnify the display of the deflections, either in all axis directions or a selected direction only, using the **Scale factor** settings on the **Deflection** tab of the **Plot Properties** dialog. Deflection results can also be animated using the animation tools. The animation illustrates the change in shape of the part from the undeflected geometry (scale factor = 0) to the final deflected geometry using the specified scale factor. The component deflection plots are useful for assessing the amount of deformation in a specific direction.

The **Examine Result** tool is particularly useful for deflection plots as it shows the coordinates of selected nodes before and after deformation and the distance between two successively selected nodes, also before and after deformation.

A warp analysis with the **Isolate cause of warpage** option activated outputs not only the total deflection results but also breaks down the total deflections according to defined contributors to warpage: differential shrinkage, orientation effects and differential cooling. Midplane and Dual Domain analyses can also display warpage due to corner effects. The contributor with the largest deflection values can be regarded as the dominant cause of warpage. Having identified the dominant cause of warpage, specific measures can then be taken to attempt to reduce the overall warpage based on the particular cause.

Deflection result

The deflection of specific sections of the part can be investigated.

Examining part displacement

The Examine result tool allows you to obtain precise warpage result values in specific model areas.

When using this tool for post-processing warpage results, the Deflection Query window appears, showing the deflection, coordinates, shrinkage, or combinations of each for the selected node(s).

TIP: You can select which warpage results are displayed in the Deflection Query window from the drop-down list.

---

1 Click **Results tab > Plots panel > Examine**.

The mouse cursor will change to small cross-hairs with a question mark.

The **Deflection Query** dialog appears.

2 Click the left mouse button once on the model location, or enter **N** followed by the node number into the **Node ID** box.

3 Click **Query**.
The window updates, displaying the selected node number, the before and after deformation X, Y, and Zvalues, and the recorded deflection at that node.

4 Optional: Click on another area or enter another node.
The dialog also shows the before and after deformation between the two points, and the percentage shrinkage.

5 Right-click the mouse button once and click **Select** to deactivate the query result function.

Viewing the warpage deflection

After a Warp analysis has been performed, it is possible to view the predicted warpage deflection for the model.

NOTE: This result does not show the actual magnitudes of deflection, only the shape that the part has adopted as a result of shrinkage differences within the part.

When assessing the warped shape of the part you should consider if the part as a whole, or just specific regions of the part has been deflected and will the warpage affect the functionality of the part (eg. assembly of mating parts).

1 Select a warpage result from the **Study Tasks** pane.

2 Click **Results tab > Animation panel** to incrementally advance the warpage visualisation.

TIP: If the deformed shape of the part is difficult to see:

1 Click **Results tab > Properties panel > Plot Properties**, and then select the **Deflection** tab.

2 Change the **Scale Factor** to a higher value.

3 Click **OK** to apply the changes, and close the dialog. As you advance the warpage visualisation the deflection is now scaled up by the factor entered.

Review the Warpage Indicator plots to identify problem areas in the part and to diagnose the cause(s) of warpage.

Isotropic shrinkage result

8

The Isotropic shrinkage result shows the average shrinkage of each element, that is, $(SH_{perpendicular} + SH_{parallel})/2$.

It is a measure of the overall level of shrinkage in the element.

A = Area after shrinkage, **B** = Material orientation direction

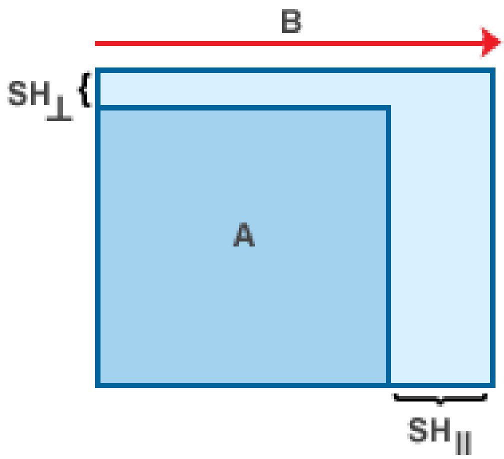

Figure 7: Isotropic shrinkage

This result uses shrinkage computed at each element before warpage is calculated, therefore these shrinkage values represent the tendency to shrink at each element before the structural influence of the surrounding part is considered.

Using this result

The overall magnitude of the shrinkage values will depend on the particular material used. Variations in isotropic shrinkage within the part indicate that warpage may occur due to differential shrinkage.

You may want to consider ways to reduce warpage due to differential shrinkage in your part.

NOTE: The Isotropic shrinkage result takes into consideration the level of crystallinity in the material throughout the part, if the material is semi-crystalline.

Material orientation results 9

The Material orientation results show the direction in which the material is oriented in the part.

This orientation will effect shrinkage, warpage and direction of impact resistance of the final part.

The orientation of the material is determined by position through part thickness, from the outer surfaces to the part center.

At the surface of the mold, the direction of flow (or shear direction) when the material first meets the surface is the probable orientation direction.

Towards the middle or core of the part, shear and surface cooling has less effect. Radial expansion can be the larger force and so material orientation perpendicular to the flow direction can occur. This will depend on part geometry, the material and any fillers used.

NOTE: To best view these results, select **Results tab > Properties panel > Plot Properties**, select the **Methods** tab, and select **Vector as segments** in the **Selection** box.

Using this result

Use this result to determine in which direction warpage will occur due to anisotropic shrinkage.

Maximum shear stress result 10

The Maximum shear stress result shows the maximum shear stress (maximum normal stress) in the part element (warpage or stress analysis), or in the wire or paddle (Microchip Encapsulation analysis) at the selected layer through the cross-section.

NOTE: This result is only available if you have selected an appropriate option in the **Stress result(s) to output** drop-down menu in the **Process Settings Wizard**.

Using this result

Consider the areas with a high maximum shear stress value and compare the results against the relevant material criteria.

Note that the plot corresponds to one particular position through the cross-section, as specified by a normalized thickness value where -1 is the bottom of the element, 0 is the centerline through the element, and +1 is the top of the element. The normalized thickness value can be viewed or modified on the **Animation** tab of the **Plot Properties** dialog.

Maximum shear stress result

Locating the maximum location of the maximum shear stress can help with the analysis of the part.

Finding the location of maximum shear stress

To find the location where maximum shear stress occurs in the model, you first need to determine when it occurs, and then configure the shear stress plot.

1. Open an analysis with results, or run an analysis.
2. Select the **Logs** check box either in the **Study Tasks** pane or below the graphics pane.

   The textual log files appear below the graphics pane in the logs pane.
3. Select the **Analysis Log** tab in the logs pane.

   A textual description of the Analysis Log appears. Next, you will find out if the maximum shear stress occurs during the filling or packing phase.

4. Scroll the Analysis Log to locate the section titled "**Filling phase results summary for the part**".
5. Note the maximum value at which the **Wall shear stress—maximum value** occurs.

   For example: `Wall shear stress—maximum (at 3.366 s) = 0.4676 MPa`
6. Scroll through the Analysis Log to locate the section titled "**Packing phase results summary for the part**".
7. Note the maximum value at which the **Wall shear stress—maximum value** occurs.
8. Compare the two values you have noted. For the highest pressure value, note the time at which it occurs (this is usually during the packing phase).
9. Select the **Shear stress at wall** check box under **Fill+Pack** in the **Study Tasks** pane.
10. Click **Step Forward** repeatedly until the result frame corresponds to the time you noted earlier on.

    The plot time is indicated underneath the plot title on the top-right corner of the screen.
11. Right-click the **Shear stress at wall** result in the **Study Tasks** pane and select **Properties**.
12. In the **Optional Settings** tab, deselect **Nodal averaged** and click **OK**.

    The location of the maximum shear stress is now assigned the highest shear stress value (colored red by default).

Mode shape result

11

The Mode shape result shows the shape of the buckled component at that load.

This is also known as eigenmode. The buckling analysis does not calculate actual displacements for such a mode shape. The mode shape is simply an arbitrarily scaled set of displacements.

Using this result

The eigenvalue associated with the buckling mode is displayed in the legend at the top right of the plot. The eigenvalue represents the factor by which the applied load, in the case of Stress analysis, or the internal loads in the case of Warp analysis, must be multiplied in order to induce buckling. If this eigenvalue is less than one the part will buckle with less than 100% of the applied load.

TIP: Click (**Step Forward**) to view each mode shape in turn if the analysis has output more than one eigenvalue.

Parallel shrinkage, before warpage result 12

The Parallel shrinkage, before warpage result indicates the percentage of shrinkage in each element in the local orientation direction, before warpage is calculated.

The local orientation direction is the direction of fiber orientation for fiber-filled materials and the direction of material orientation (flow direction before freezing) for all other materials.

This result is produced by a Warp analysis based on predicted shrinkages from the specified material shrinkage model (residual stress, CRIMS, or residual strain).

This result uses shrinkage computed at each element before warpage is calculated, therefore these shrinkage values represent the tendency to shrink at each element before the structural influence of the surrounding part is considered.

NOTE: This result is not available for 3D analysis technology.

Using this result

Use this result to find out whether differential shrinkage is likely to cause warpage. Differential shrinkage may be caused by differences in shrinkage values between different regions of the part, or by large differences between the parallel and perpendicular shrinkage.

This result is not displayed by default. To display this result, add a new plot to the display in the active study or customize the default results list to include this result.

Things to look for

Look for regions in which there is high differential shrinkage. These regions are more likely to deform.

Phase Shift for light coming from +/- Z direction result 13

The Phase Shift for light results display the difference (expressed as fractions of a wavelength, where 1 cycle is 360°) between the phases of horizontally-polarized and vertically-polarized light as it passes through the part from the direction named in the result. A wavelength of 546.1 nm (green) is assumed.

These results are created at the end of a 3D Warp analysis.

NOTE: This result is generated only if you have selected the **Birefringence analysis if material data includes optical properties** checkbox in the Fill+Pack page of the **Process Settings Wizard** and the material has measured optical properties.

Using this result

The phase shift indicates how out-of-step the phases of horizontally-polarized and vertically-polarized light became by passing through the part. This becomes important if the light was initially polarized, or subsequently passes through another optical element which further polarizes the light. If so, the transmitted light may vary in brightness or exhibit colored bands.

The meaning of “horizontal” and “vertical” depends on the orientation of the light source. They are mutually orthogonal planes of polarization, the intersection of which contains the incident light beam, for each location on the part. The value displayed is for the planes which maximize the phase shift, i.e., the worst case.

This result is meaningful only on the sides of the part not facing the light source. The result is absent for sides of the part facing the light source because there has been no phase shift at this point.

TIP: Use the plot properties to decrease the scaling to show only the areas that do not meet specifications, by setting the lower limit to 90° or the part's specified maximum tolerance.

Things to look for

Values of more than 90° significantly affect the polarization of the transmitted light. A tolerance of 30° or even less is advisable in optically important sections of the part.

NOTE: Create a custom plot if you want to see results for a different wavelength.

---

14 Perpendicular shrinkage, before warpage result

The Perpendicular shrinkage, before warpage result indicates the percentage of shrinkage in each element in the plane of the element, at right angles to the local orientation direction, before warpage is calculated.

The local orientation direction is the direction of fiber orientation for fiber-filled materials and the direction of material orientation (flow direction before freezing) for all other materials.

This result is produced by a Warp analysis based on predicted shrinkages from the specified material shrinkage model (residual stress, CRIMS, or residual strain).

This result uses shrinkage computed at each element before warpage is calculated, therefore these shrinkage values represent the tendency to shrink at each element before the structural influence of the surrounding part is considered.

NOTE: This result is not available for 3D analysis technology.

Using this result

Use this result to find out whether differential shrinkage is likely to cause warpage. Differential shrinkage may be caused by differences in shrinkage values between different regions of the part, or by large differences between the parallel and perpendicular shrinkage.

This result is not displayed by default. To display this result, add a new plot to the display in the active study or customize the default results list to include this result.

Things to look for

Look for regions in which there is high differential shrinkage. These regions are more likely to deform.

15 Retardation for light coming from +/- Z direction result

The Retardation for light results display the absolute difference (as a length) between the phases of horizontally-polarized and vertically-polarized light as it passes through the part from the direction named in the result.

These results are created at the end of a 3D Warp analysis.

NOTE: This result is generated only if you have selected the **Birefringence analysis if material data includes optical properties** checkbox in the Fill+Pack page of the **Process Settings Wizard** and the material has measured optical properties.

Using this result

The retardation indicates how out-of-step the phases of horizontally-polarized and vertically-polarized light became by passing through the part. This becomes important if the light was initially polarized, or subsequently passes through another optical element which further polarizes the light. If so, the transmitted light may vary in brightness or exhibit colored bands.

The meaning of “horizontal” and “vertical” depends on the orientation of the light source. They are mutually orthogonal planes of polarization, the intersection of which contains the incident light beam, for each location on the part. The value displayed is for the planes which maximize the phase difference, i.e., the worst case.

This result is meaningful only on the sides of the part not facing the light source. The result is absent for sides of the part facing the light source because there has been no phase shift at this point.

TIP: Use the plot properties to decrease the scaling to show only the areas that do not meet specifications, by setting the lower limit to the part's maximum specified tolerance.

Things to look for

Values larger than 0.25 of the incident light wavelength significantly affect the polarization of the transmitted light. A tolerance of 0.1 wavelength or even less is advisable in optically important sections of the part.

Strain in first principal direction result

16

The Strain in first principal direction result shows the strain in first principal direction values throughout the part, in the laminate through the thickness of the part shown by the Normalized Thickness value in the legend, and the load factor shown in the legend.

In the case of a Warp analysis, the values represent *residual strains*, which are the actual strains minus the strains due to pure shrinkage.

NOTE: This result is only available if you have selected an appropriate option in the **Stress result(s) to output** drop-down menu in the **Process Settings Wizard**.

Using this result

Note that the plot corresponds to one particular position through the cross-section, as specified by a normalized thickness value where -1 is the bottom of the element, 0 is the centerline through the element, and +1 is the top of the element.

The normalized thickness value can be viewed or modified on the **Animation** tab of the **Plot Properties** dialog.

This result displays the strain distribution within the part as a contour plot. You can also display the strains in a tensor plot using the Strain tensor result.

Strain in second principal direction result

17

The Strain in second principal direction result shows the strain in second principal direction values throughout the part, in the laminate through the thickness of the part shown by the Normalized Thickness value in the legend, and the load factor shown in the legend.

In the case of a Warp analysis, the values represent *residual strains*, which are the actual strains minus the strains due to pure shrinkage.

NOTE: This result is only available if you have selected an appropriate option in the **Stress result(s) to output** drop-down menu in the **Process Settings Wizard**.

Using this result

Note that the plot corresponds to one particular position through the cross-section, as specified by a normalized thickness value where -1 is the bottom of the element, 0 is the centerline through the element, and +1 is the top of the element.

The normalized thickness value can be viewed or modified on the **Animation** tab of the **Plot Properties** dialog.

This result displays the strain distribution within the part as a contour plot. You can also display the strains in a tensor plot using the Strain tensor result.

18 Strain tensor (warp) result

The Strain tensor result shows the strain in the selected direction (default = first principal direction) throughout the part, in the laminate through the thickness of the part shown by the Normalized Thickness value in the legend, and the load factor shown in the legend.

In the case of a Warp analysis, the values represent *residual strains*, which are the actual strains minus the strains due to pure shrinkage.

NOTE: This result is only available if you have selected an appropriate option in the **Stress result(s) to output** drop-down menu in the **Process Settings Wizard**.

Using this result

Note that the plot corresponds to one particular position through the cross-section, as specified by a normalized thickness value where -1 is the bottom of the element, 0 is the centerline through the element, and +1 is the top of the element.

The normalized thickness value can be viewed or modified on the **Animation** tab of the **Plot Properties** dialog.

You can display strains in the second principal direction, or one of the 6 tensor components, using the options provided on the **Tensor** tab of the **Plot Properties** dialog.

This result displays the strain distribution within the part as a tensor plot. You can also display the strains as a contour plot using either the Strain in first principal direction or Strain in second principal direction results.

19 Stress tensor (warp) result

The Stress tensor result shows the stress in the selected direction (default = first principal direction) throughout the part at ejection, in the laminate through the thickness of the part shown by the Normalized Thickness value in the legend, and the load factor shown in the legend.

NOTE: This result is only available if you have selected an appropriate option in the **Stress result(s) to output** drop-down menu in the **Process Settings Wizard**.

Using this result

Consider the principal stresses when the material is brittle. Consider the areas with a high level of stress and compare the results against the relevant material criteria. If the absolute value of the second principal stress of an element is greater than the relevant material criteria then the part will fail.

Note that the plot corresponds to one particular position through the cross-section, as specified by a normalized thickness value where -1 is the bottom of the element, 0 is the centerline through the element, and +1 is the top of the element.

The normalized thickness value can be viewed or modified on the **Animation** tab of the **Plot Properties** dialog.

You can display stress in the second principal direction, or one of the 6 tensor components, using the options provided on the **Tensor** tab of the **Plot Properties** dialog.

This result displays the stress distribution within the part as a tensor plot. You can also display the stresses as a contour plot using either the Stress in first principal direction or Stress in second principal direction results.

Warp analysis log

20

The Warp analysis log contains a record of the analysis.

This log can be accessed from the **Study Tasks** pane at any time during or after the analysis.

Analysis Progress Table

The following columns are displayed in the Warp analysis log progress table:

Kstep	The non-linear analysis load step number
Kstra	Indicates the strategy used for the solution
Nref	The number of reformations of the stiffness matrix at each step
Nite	The number of iterations in each step
Ipos	Degrees of freedom
Negpv	This is the negative diagonal term in stiffness matrix after LDL decomposition.
Detk	This is a relative measure of the stiffness of the structure. This value is always one on the first step, so values larger than one indicate increasing stiffness.
Rfac	The fraction of load applied at a step. It is important to check that the last step is 1.
Displacement	Indicates the node displacement in the Z-direction at each step.

Convergence Failure

Sometimes the following messages can occur in the progress table:

* CONVERGENCE FAILURE *

or

* DIVERGENCE OCCURRED *

When you see these warnings, look at the second column from the left (Kstra), which shows the different strategies used during the analysis. If the last value of Kstra in the progress table is in the range 0-4, then the solution will be satisfactory. If it is 5, this indicates that the Structural analysis program has given up equilibrium iterations, and the results will probably be unreliable.