Autodesk® Moldflow® Insight 2012

AMI Stress Analysis Results

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

Stress analysis results

1

This help topic specifies the results for a Stress analysis on a thermoplastic material.

Text based results

The following table lists the text results generated for a Stress analysis.

Results
Analysis log
Results Summary
Analysis Check

Graphical results

The following table lists the graphical results that are created for a Stress analysis 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 type
Stress in first principal direction result on page 2	
Stress in second principal direction result on page 3	
Stress, Mises-Hencky result on page 4	
Von-Mises stress, core result on page 5[1]	
Von-Mises stress, maximum result on page 6[1]	
Stress, Normal to Core result on page 7[1]	

[1] This result is output only when the option to **Perform core shift analysis** is selected in the solver parameters and a core is included in the model.

Stress in first principal direction result

2

The Stress in first principal direction result shows the first principal stress (maximum normal stress) in the part (Warp or Stress analysis), or in the wire or paddle (Microchip Encapsulation analysis), at the selected position through the cross-section, after ejection.

Positive values correspond to tension in the part, and negative values correspond to compression.

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 first principal stress result 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 first 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.

Also consider the second principal stress when the material is brittle. It is possible for the absolute value of the second principal stress to be greater than the absolute value of the first principal stress if an element is in compression.

Things to look for

- Always use the maximum value of stress when checking to see if the part will fail.

Stress in second principal direction result

3

The Stress in second principal direction result shows the second principal stress in the part (Warp or Stress analysis), or in the wire or paddle (Microchip Encapsulation analysis), at the selected layer through the cross-section, after ejection.

Positive values correspond to tension in the part, and negative values to compression.

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**.

NOTE: The direction of the second principal stress (minimum normal stress) on the top/bottom of each element is perpendicular to the direction of the first principal stress.

Using this result

Consider the second principal stress 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.

Also consider the first principal stress when the material is brittle.

Things to look for

- Always use the maximum value of stress (either top or bottom result) when checking to see if the part will fail.

Stress, Mises-Hencky result 4

The Stress, Mises-Hencky result shows the Mises-Hencky stress (maximum normal stress) in the part (Warp or Stress analysis), or in the wire or paddle (Microchip Encapsulation analysis), at the selected layer through the cross-section, in the deformed state after ejection has occurred.

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

Use this result when the material is ductile. Consider the areas with a high level of stress 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.

NOTE: Due to the extrapolation from elemental centroids to the edges of elements, it is possible for small negative values to be generated for this result (Mises-Hencky stress values must, by definition, be positive). These small negative values should be taken to equal zero.

Things to look for

- Always use the maximum value of stress (either top or bottom) when checking to see if the part will fail.

Von-Mises stress, core result

5

The von-Mises stress, core result shows the von-Mises stress in the core at ejection.

This result is output from a Core-shift Warp or Stress analysis. Use this result when the material is ductile.

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**.

Areas with high levels of stress shows increased areas of fatigue on the core, which can lead to the core cracking and the part failing.

Using this result

You can use the von-Mises stress, core result to ensure the areas with the maximum stress are below the relevant material criteria. Ensuring that the von-Mises stress, core values are low overall decreases the chance of part failure.

Von-Mises stress, maximum result

6

The von-Mises stress, maximum result shows the maximum von-Mises stress in the part at ejection.

The von-Mises stress, maximum result is output from a Core-shift Warp or Stress analysis. Use this result when the material is ductile.

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**.

Areas with high levels of stress shows increased areas of fatigue, which can lead to the part cracking and failing.

Using this result

You can use the von-Mises stress, maximum result to ensure the maximum stress is below the relevant material criteria. Ensuring that the von-Mises stress, maximum values are low overall decreases the chance of part failure.

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.

NOTE: Due to the extrapolation from elemental centroids to the edges of elements, it is possible for small negative values to be generated for this result. These small negative values should be taken to equal zero.

Stress, Normal to Core result 7

The Stress, Normal to Core result shows the value most cavity pressure sensors report as the "cavity pressure". While the polymer is molten the stress value is very close to the cavity pressure but when the polymer is frozen and the mold (core) has a significant elastic deformation then the value may be significantly different from the pressure result.

NOTE: This result is only available for Dual Domain or Midplane meshes, and requires that you enable the **Core shift** analysis in the **Solver parameters** of the **Process Settings Wizard**.

Using this result

Use this result along with the Pressure result, to see the effect of the mold elasticity on the values when validating prediction of the experimental cavity pressure results. High local normal stresses caused by the core elasticity can also cause some cracking in the moldings if the plastic is brittle

You can also use the Stress, Normal to Core result along with the Residual Stress result for the prediction of cracking.

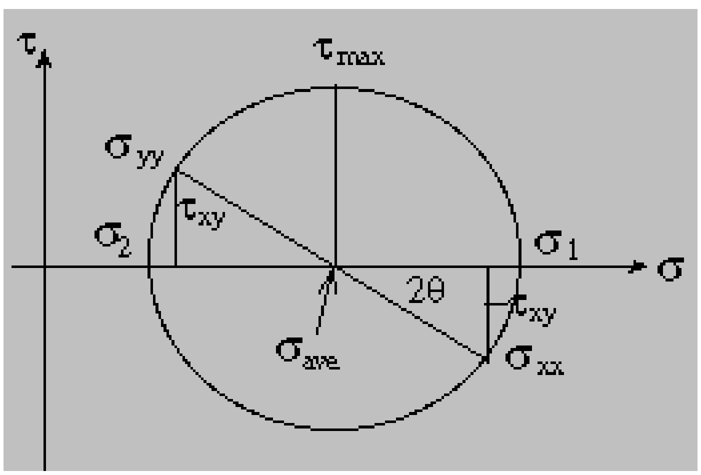
Derivation of stress results 8

For a Midplane Stress or Warp analysis, the Stress Settings or Warp Settings page of the Process Settings Wizard includes an option to select which types of stress results to output.

You can select one or more of the following results:

- Principal residual stress (first principal and second principal stresses).
- Maximum shear stress.
- Mises-Hencky stress.

The mathematical derivation of these results can be illustrated with the aid of the well known Mohr's circle which depicts the stress state at a point.



First principal and second principal stress

The principal stresses are the extreme values of the normal stresses. Since they characterize the physical state of the stress at a point, they are independent of any coordinates of reference. They are calculated in the following way:

$$\sigma_1 = \frac{\sigma_{xx}+\sigma_{yy}}{2} + \sqrt{\left(\frac{\sigma_{xx}-\sigma_{yy}}{2}\right)^2 + \tau_{xy}^2} \quad \sigma_2 = \frac{\sigma_{xx}+\sigma_{yy}}{2} - \sqrt{\left(\frac{\sigma_{xx}-\sigma_{yy}}{2}\right)^2 + \tau_{xy}^2}$$

where σ_1 is the maximum normal stress and σ_2 is the minimum normal stress. The corresponding directions of the principal stresses are called the first principal and second principal direction respectively. The angle for these is calculated using the following equation:

$$\tan 2\theta = \frac{2\tau_{xy}}{\sigma_{xx}-\sigma_{yy}}$$

NOTE: Positive principal stress values represent tension and negative values compression.

Maximum shear stress

The Maximum shear stress is the extreme value of the shear stress and its value is calculated as follows:

$\tau_{max} = \sigma_1 - \sigma_2$

Mises-Hencky stress

The Mises-Hencky stress is calculated as follows:

$\sigma_{MH} = \sigma_1^2 - \sigma_1\sigma_2 + \sigma_2^2$

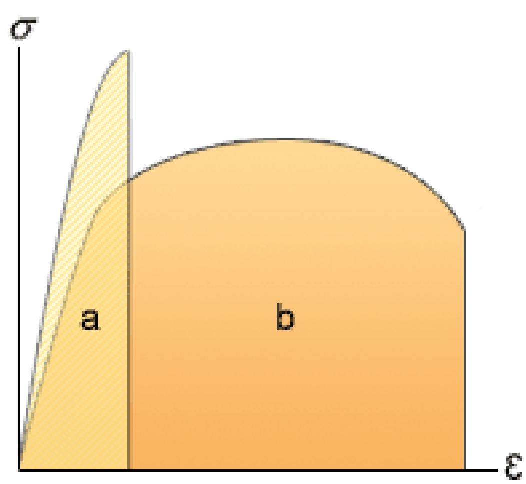
NOTE: As can be seen in the above equation, Mises-Hencky stress values will always be positive. Due to the extrapolation from elemental centroids to the edges of elements, it is possible for small negative values to be generated for Mises-Hencky stress. These small negative values should be regarded as equal to zero.

Interpreting the stress results

In general, when examining stress results, check the distribution of stresses within the part and the maximum stress levels in the part.Stress and Warp outputs results for both the top and bottom of the element (Normalized thickness = 1 and -1 respectively).

These need to be compared against recommended maximum stresses for the material and any relevant design criteria for the part, for example, specified failure criteria.

Non-filled, isotropic materials will in general exhibit either brittle or ductile behavior, as illustrated in the following figure, where (a) represents brittle and (b) ductile stress-strain behavior.



The recommended stress results to consider in each case are:

- For brittle materials, consider the principal stress results.
- For ductile materials, consider the Mises-Hencky result.

For fiber-filled, anisotropic materials, the behavior of the part under load, the mechanics of failure, and the design criteria for failure will be considerably more complex than for an isotropic material. Stress analysis of composite materials, and interpretation of the results obtained, requires special expertise on the part of the user.

Always run a Fill+Pack analysis to generate fiber orientation data prior to a Stress or Warp analysis. Both the fiber orientation and Stress/Warp analyses output results on a per-laminate basis through the thickness of the part.

NOTE: Principal stress orientation may not correspond to principal fiber orientation. Stress orientation data is dependent on the stress state and is located in the center of each layer. Fiber orientation data is calculated at the layer interfaces and is a material property.
