Autodesk® Moldflow® Insight 2012

AMI Shrink Analysis

Revision 1, 22 March 2012.

Contents

Shrink analysis

1

Shrink analysis enables you to determine an appropriate shrinkage allowance to use to cut the mold taking into account the shrinkage characteristics of the material being used to mold the part and the molding conditions.

Every part which is injection molded requires someone to select the dimensions to which the mold must be cut. In the past, many precision parts have required molds to be heavily modified so that tolerances can be met successfully. On some occasions, molds have been scrapped several times over, in order to achieve the required dimensions, incurring huge costs and significant delays in time to market for the product.

How it works

The key features of the Shrink analysis are:

- Calculation of a recommended shrinkage allowance.
- Graphical display indicating whether it is valid to apply this single shrinkage allowance value across the part.
- Optional definition of critical dimensions and their associated tolerances. Where critical dimensions are defined, the Shrink analysis predicts whether the specified tolerances can be met if the recommended shrinkage allowance is used, included detailed dimensional and tolerance information resolved into X, Y and Z directions.

Material shrinkage is defined as the reduction in the size of a molded component in any direction after it has been ejected from the mold. It is related to the flow and cooling conditions under which the component is injection molded. Shrinkage data characterizes this reduction in component size due to shrinkage for a range of different processing conditions. Shrink analysis is available for any material in the Materials Database that has been shrinkage characterized.

NOTE: Shrinkage is affected greatly by fiber orientation.

Although it is possible to run a Shrink analysis with a fiber-filled material without selecting the Fiber-orientation analysis option, the result will not take orientation into account and will not be as accurate. If you are analyzing a fiber-filled material you should turn on the fiber analysis option in the Fill+Pack process settings.

TIP:

If your material contains anisotropic matrix material properties, warpage and shrinkage values may be more accurately calculated using the Mori-Tanaka micomechanical model.

Shrink analysis

You can determine an appropriate shrinkage allowance by running a Shrink analysis.

Setting up a Shrink analysis

The following table summarizes the setup tasks required to prepare a Shrink analysis of a non fiber-filled, or fiber-filled thermoplastic material.

Setup task	Analysis technology
Fill analysis	
Selecting a material	

Optional setup tasks

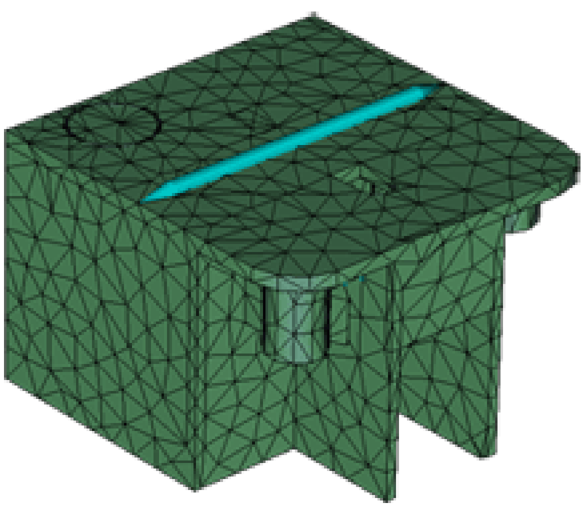
Setup task	Analysis technology
Critical dimensions on page 3	

2 Critical dimensions

Shrink analysis calculates a recommended shrinkage allowance to be used when designing the mold taking into account the shrinking characteristics of the current material as obtained from shrinkage testing.

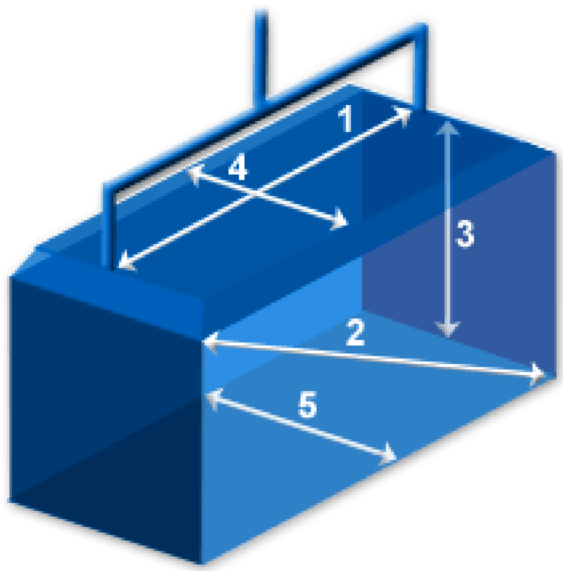
Setting dimensions

A part drawing will often indicate tolerances for certain dimensions of the part. By critical dimensions we mean those dimensions where the part specifications demand that the indicated tolerances be met. Critical dimensions, and the specific tolerances associated with them, can be defined in Autodesk Moldflow Insight using the **Set critical dimensions** tool in the **Analysis** menu. As each critical dimension is defined, it is displayed as a double ended arrow on the part model.



If one of more critical dimensions have been identified, then the Shrink analysis will verify whether these critical dimensions can be met given the calculated recommended % shrinkage allowance for the part as a whole. This information is displayed in the analysis log. You can use this information to compare the part dimensions with the mold dimensions required to produce a finished part within tolerance.

The following picture illustrates how critical dimensions might be defined for a box-shaped part.



Critical dimensions

Critical dimensions are those dimensions where the part specifications demand that the indicated tolerances be met.

Setting critical dimensions

Shrink analysis will verify whether these critical dimensions can be met given the calculated recommended % shrinkage allowance for the part as a whole.

1. Click  **Boundary Conditions tab > Shrink panel > Critical Dimensions**.
   The **Critical Dimensions** dialog appears.
2. Check whether model nodes are displayed. If not, in the **Layers** pane, select the check box next to the layer(s) containing the model nodes.
3. For each critical dimension to be defined:
   a. Click on the node location at one end of the critical dimension. The number of the selected node will be displayed in the **Node 1** box.
   b. Click on the node location at the other end of the critical dimension. The number of the selected node will be displayed in the **Node 2** box and the distance between the two nodes is displayed.
   c. Specify the allowed +ve and -ve tolerances for the selected model dimension or accept the default values which represent +/- 5% of the modeled dimension.
   d. Click **Apply** to create the critical dimension input for Shrink analysis. The critical dimension will appear as a light blue bar on the part.
4. To change the attributes of an existing critical dimension:

a Click (**Select**).
b Click on the critical dimension symbol (light blue solid line).
c Right-click and select **Properties** from the context sensitive menu.
d Edit the properties as required and click **OK**.

Critical dimensions

Use this dialog to set critical dimension for a Shrink analysis.

Critical Dimension dialog

This dialog is used to set the properties associated with a critical dimension for Shrink analysis.

To access this dialog to edit the properties of an existing critical dimension, select a critical dimension symbol in the model pane, then either select **Geometry tab > Properties panel > Edit**, or press **Alt-Enter**, or right-click and select **Properties**.

The set of property values defined by the dialog are saved to a property set with the description shown in the **Name** box.

Set Critical Dimensions tool

This tool is used to define a critical dimension, and its associated dimensional tolerances, for shrinkage analysis.

To access this panel, select **Boundary Conditions tab > Shrink panel > Critical Dimensions**.

Shrinkage models

3

There are several shrinkage models used in this product, depending upon how much information about the material is available.

The following options are available:

Uncorrected residual stress	This option will be selected when there is no shrinkage data available for the material. In this case, the Fill+Pack analysis predicts the residual stress values within in the part, based on the flow and thermal history during the molding cycle.
Corrected residual in-mold stress (CRIMS)	This option is the default when shrinkage testing has been performed on the material. This model is the most accurate because it is obtained by correlating actual tested shrinkage values with Fill+Pack analysis predictions.
Residual strain	This option is selected for those materials where the CRIMS model was found to not adequately describe the shrinkage behavior of the material.

Residual strain shrinkage prediction method

The residual strain shrinkage prediction method is the older of the two shrinkage prediction methods employed by Autodesk Moldflow in its Warp analysis product.

The residual strain method is based on the following empirical model for shrinkage:

S⊠a1⊠MV+a2⊠Mc+a3⊠Mo⊠a4⊠Mr+a5
S⊠a6⊠MV+a7⊠Mc+a8⊠Mo⊠a9⊠Mr+a10

where

- S⊠and S⊠are the predicted values of linear shrinkage parallel and perpendicular to the direction of flow respectively,
- a1,…,a10 are constants for a given material,
- MV is a measure of the volumetric shrinkage,
- Mc is a measure of the crystallization,
- M0⊠ M0⊠are measures of the molecular orientation parallel and perpendicular to the direction of flow, and
- Mr is a measure of mold restraint.

The coefficients a1,…,a10 are constants for a given material and are determined by means of a shrinkage characterization procedure whereby shrinkage data obtained experimentally from molding a standard test piece are fitted to the above equations.

The various measures in the model, volumetric shrinkage, crystallization, material orientation and mold restraint, are calculated by the Fill+Pack analysis for a given warpage simulation. These measures are described in more detail in the following sections.

The above shrinkage model has been extended to account for bending moments due to temperature differences from one side of the mold to the other, as determined from Cool analysis. Incorporation of these effects gives shrinkages parallel and perpendicular to the flow direction on the top and bottom of each element. This aspect of the model is described in the section Mold Restraint Term below.

Volumetric Shrinkage Term

Volumetric shrinkage is a fundamental part of the shrinkage calculations. The main factors affecting volumetric shrinkage are holding pressure and the temperature history of the melt. The volumetric shrinkage for each element is calculated from the "equilibrium" pvT relationship for the material, using the temperature/pressure history experienced during packing and cooling.

Volumetric shrinkage is calculated as: VS=100⊠–vvfrz where vfrz is specific volume of polymer at the time when either the polymer in element becomes completely frozen or melt pressure in the element becomes atmospheric; v is the specific volume of polymer at atmospheric pressure and room temperature. Specific volume at a particular pressure and temperature is calculated using the pvT relationship.

Crystallization Term

Equilibrium pvT data is not sufficient for describing the volumetric shrinkage of semi-crystalline materials. The amount of volumetric contraction that occurs in these materials also depends on the degree of crystallization. The degree of crystallization in the part is primarily affected by the mold temperature. The shrinkage calculation uses the crystallization kinetics of the material to determine the volume contraction due to crystallinity levels.

Crystallization is a function of both temperature and time. The level of crystallization is determined by cooling rates. Rapid cooling rates are associated with lower levels of crystalline content and vice versa.

In injection molded parts, thick regions tend to cool slowly relative to thinner sections and so have higher crystalline content and hence higher volumetric contraction. On the other hand, thin regions cool very quickly and so have lower crystalline content and hence lower volumetric contraction than that predicted from equilibrium pvT data.

Orientation Term

During shear flow, polymer molecules align themselves in the direction of flow. The extent of this orientation depends on the shear rate to which the material is subjected and the temperature of the melt.

When the material stops flowing, the induced molecular orientation begins to relax at a rate dependent on the material's relaxation time. If the material freezes before relaxation is complete, the molecular orientation is “frozen-in”. This frozen-in orientation will result in different levels of shrinkage in the directions parallel and perpendicular to the direction of material orientation.

To determine the level of frozen-in orientation, the Fill+Pack analysis calculates the following quantities for each element and each grid point i across that element at the time when the grid point freezes:

- The shear stress, □
- The cooling rate, dT/dti
- The flow angle, □ relative to the element's local X-axis. The local X-axis is the direction from the first node number to the second number in the element's definition. Note that the flow direction can be different from laminate to laminate because the growth of frozen layer may change the flow channel and hence the flow direction.

The final level of “frozen in” orientation is determined by taking the molecular orientation level at the time the material stops flowing, which is proportional to the shear stress at that time, and reducing the level of orientation by an amount determined by the relaxation characteristics of the material, which is a function of the cooling rate.

Having thus determined the degree of orientation, □ at each grid point, the orientation measure, □x, at grid point *i* and in the direction parallel to the local X-axis is then given by:

□x=□□cos□

The orientation measure in the perpendicular direction, □y, is determined in a similar way.

The measures of orientation, parallel and perpendicular to the local X-axis for the element as whole, □ and □ respectively, is then determined by summing the grid point measures, that is,

□=∑i=1n□x □=∑i=1n□y

where *n* is the number of grid points in the plastic.

The elemental material orientation direction, relative to the local X-axis, □ is then defined to be:

□arctan□□

Mold Restraint Term

While the part is in the mold, it is assumed that it cannot physically contract in the plane of the element. However, contraction in the thickness direction is allowed.

As the material contracts, residual stresses build up in the part. The temperature history of a plastic element during the filling, packing and cooling phases affects the rate at which these stresses can relax. A measure of mold restraint is calculated by adding contributions from a number of small temperature increments, in which the rate of relaxation is determined from the current temperature.

Equivalent Thermal Strains

The residual strain shrinkage model described above gives, for each element, a value of parallel and perpendicular shrinkage that represents an average value through the thickness of that element. In reality the level of shrinkage varies through the thickness. If this shrinkage distribution is asymmetric about the cavity centerline, a bending moment is created which may influence the warpage of the part.

If the temperature distribution across the element thickness is known, the shrinkage distribution in the thickness direction, *SH(z)* , can be approximated as follows:

$$SH(z) = (T(z) - T_{room}) \times \frac{SH}{T_{av} - T_{room}}$$

Where:

T(z) is the temperature distribution in the plastic when the center freezes, that is, when the element is fully frozen, as obtained from Cool analysis. The peak value of *T(z)* is the freeze temperature of the material and will be located at a position across the element thickness as determined from the Cool analysis. In evaluating the above equation, the temperature at each mold-cavity interface is approximated to be the end of cycle mold temperature and the temperature distributions either side of the maximum temperature are approximated by parabolic curves.

T_{av} is the average of *T(z)* over the thickness,

SH is the average shrinkage as predicted by the shrinkage model,

SH(z) is an "effective" coefficient of thermal expansion. It is not a true thermal expansion coefficient because it includes the effects of other shrinkage processes, e.g. crystallinity.

This *SH(z)* distribution is then linearized, i.e. converted into a straight line, in such a way as to preserve the bending effect of the distribution. The bending effect is characterized by the integral:

$$\frac{12}{h^3} \int_{z=-h/2}^{z=h/2} z\, SH(z)\, dz$$

where *h* is the element thickness.

Note that this conversion to a straight line is not an approximation but a necessity, because the elements types supported by the Autodesk Moldflow stress analysis program (like most stress analysis programs) can only handle a linear distribution of strain through the thickness of the element.

The result of this linearization is a straight line shrinkage distribution, *SHL(z)*. This is converted back to a linearized temperature distribution *TL(z)* using the above equations. The reason for converting back to a temperature distribution is historical; ABAQUS only accepts coefficients of thermal expansion and temperature change, not direct initial strain input.

The top and bottom temperature of the element, as determined from *TL(z)*, the room temperature, and the ⊠values for the parallel and perpendicular directions are saved as input for the Warp analysis. In the Warp analysis, *SHL(z)* is reconstructed from these values using the two equations at the start of this section.

Residual strain shrinkage prediction method

The residual strain shrinkage prediction method is the older of the two shrinkage prediction methods employed by Autodesk Moldflow in its Warp analysis product.

Changing the default shrinkage model

If a material has been shrinkage tested, then the Autodesk Moldflow materials database includes either the Residual strain shrinkage model or the Corrected residual in-mold stress (CRIMS) shrinkage model, depending on which provides the best match between predicted and actual measured shrinkages. If the material has not been tested for shrinkage, then the CRIMS model is not available, and the uncorrected residual stress model is used instead.

The selected shrinkage model is shown on the **Shrinkage** tab of the **Thermoplastics Material** dialog. You can however change the shrinkage model using the following procedure.

1. Click **Home tab > Molding Process Setup panel > Process Settings**, or double-click the process settings icon in the **Study Tasks** pane.

   The **Process Settings Wizard** appears.

2. Click **Advanced options**, on the Fill Settings or Fill+Pack Settings page of the Wizard, to display the **Fill+Pack Analysis Advanced Options** dialog.
3. Click **Edit** next to the Molding Material drop-down menu, to display the **Thermoplastics Material** dialog.
4. Click the **Shrinkage Properties** tab, then select the required shrinkage model in the **Shrinkage model** drop-down list.

   If an **Edit model coefficients** button appears, you can click on it to view further information about the shrinkage model coefficients, the shrinkage testing performed, or additional settings like the Use CRIMS option.

Residual strain shrinkage prediction method

Use this dialog to show the coefficients for residual strain shrinkage obtained from a Shrink analysis.

Residual Strain Model Coefficients dialog

This dialog shows the coefficients for the Moldflow proprietary Residual Strain shrinkage model, as obtained from shrinkage testing using the default Flow/Fiber data set.

Residual Strain Model Coefficients for RSC Fiber model dialog

This dialog shows the coefficients for the Moldflow proprietary Residual Strain shrinkage model, as obtained from shrinkage testing using the RSC Fiber model data set.

Residual stress shrinkage prediction method

The residual stress shrinkage prediction model has been formulated assuming linear thermo-viscoelastic material behavior. It accounts for the stress developed while the material cools under pressure in the mold. In this method, rather than calculate shrinkage strain we directly calculate a residual stress distribution for each element.

The residual stress distribution provides the stress across the thickness of each element in directions parallel and perpendicular to flow. This stress distribution is then input to the Stress analysis program to obtain the deflected shape of the part. If in addition experimental shrinkage data is available for the material, considerably more accurate prediction of shrinkages and hence part deflections can be obtained than using the residual strain method.

General description of the method

The model has been formulated assuming linear thermo-viscoelastic material behavior. It accounts for the stress developed while the material cools under pressure in the mold. In particular, the model accounts for the thermally induced stress which arises from freezing and subsequent shrinkage of the material as well as the pressure induced stress. The latter stress is caused by the action of the melt pressure on the solidified material forming the frozen layer. Being theoretically based, this model has the advantage that it can be used even if no shrinkage data is available for the material. However, its performance can be improved greatly when shrinkage data is available.

The prediction of shrinkage and warpage is based on the computed thermally and pressure induced residual stress distribution. The computational procedure in the present development is listed below. This is for fiber-filled materials. For unfilled materials the procedure is similar, but there is no need for the mechanical property calculation.

For each time step:

1 Calculate fluid mechanics:

- Pressure p, Flow rate Q...
- Fiber orientation

2 Calculate heat transfer:

- Temperature T
- Frozen layer

3 Calculate thermodynamics:

- $f(p,v,T)=0$

4 Update viscosity.

5 Does the solution converge?

- If NO, repeat from Step 1.
- If YES, go to Step 6.

6 Calculate micro-mechanics:

- Thermo-mechanical properties:

  $C_{ijkl}=f(\phi, L/d, a_{ij}, a_{ijkl})$

7 Calculate thermoviscoelasticity:

- Thermally and pressure induced stresses

8 IF $T<T_{ejection}$, go to the next time step (repeat from Step 1).

Note: When the Residual Thermal Stress shrinkage model is selected, the Warp analysis uses the asymmetric information produced by the asymmetric Fill+Pack analysis. The asymmetry could be caused by temperature differences between the two sides of the mold or/and by branching geometry. Therefore, even if there is no Cool analysis result, the warpage results could show some asymmetric effects if the part has branching geometry. However, if the Fill+Pack analysis is symmetric, then the Warp analysis will be symmetric and therefore ignore differential cooling.

Application of this method when no shrinkage data is available

A general form of the anisotropic stress-strain relation for linear thermoviscoelasticity may be written as $\sigma_{ij}=\int_{-\infty}^{t} c_{ijkl}(\xi-\xi')\frac{\partial \varepsilon_{kl}}{\partial t'}dt' - \int_{-\infty}^{t}\beta_{ij}(\xi-\xi')\frac{\partial T}{\partial t'}dt'$ where:

- C_{ijkl} and β_{ij} are tensors defining the mechanical and thermal characteristics of the material, respectively.
- ξ is a pseudo-time scale which accounts for the temperature dependence of the material and is defined by $\xi=\int_0^t \frac{1}{a_T}dt'$ where a_T is the time-temperature shift factor characterized by either the WLF equation or the Arrhenius equation depending on the material and the temperature range.

The thermo-viscoelastic model has been found to be extremely sensitive to material data and indeed material data is not readily available for its use. In the absence of experimentally obtained viscoelastic data, we have utilized a viscous-elastic model with two forms:

- For non-fiber-filled materials, the model is isotropic whereby the mechanical property tensor C_{ijkl} is defined by the modulus and Poisson's ratio of the material, as stored in the Autodesk Moldflow materials database, and the thermal expansion coefficients used to define the thermal property tensor α are obtained from the pvT data for the material.
- For fiber-filled materials, these tensors are defined using anisotropic mechanical and thermal properties for the composite material, which are calculated from the fiber orientation distribution obtained from the Fill+Pack analysis. In this case the model predicts stresses along and transverse to the fiber orientation direction.

The viscous-elastic model is based on the following assumptions:

- There is no stress build up in the material until the material is below the transition temperature.
- With respect to the local element coordinates in which the x3 direction is normal to the local midplane, the shear stresses $\sigma_{13} = \sigma_{23} = 0$.
- The normal stress σ_{33} is constant across the thickness.
- As long as $\sigma_{33} < 0$, the material sticks to the mold walls.
- A constrained quench condition is prescribed in all the cases as long as the material is in the mold.
- Mold elasticity is neglected. (Elasticity of mold cores is taken into account in a core shift analysis.)
- The material behaves as an elastic solid after the part is ejected.

This model is available for use with all materials on the Autodesk Moldflow materials database, whether they have undergone shrinkage characterization or not. In the case of fiber-filled materials, the model requires that the option to perform **Fiber orientation analysis if fiber material** be selected in the **Process Settings Wizard—Fill+Pack Settings** dialog.

This model is capable of predicting shrinkage trends but can have substantial errors with regard to the absolute values obtained. It is useful for designing to reduce warpage but absolute values are less accurate than when shrinkage data is available. For unfilled materials, the model is isotropic as there is no facility to calculate the effects of molecular orientation or crystallinity in the flow analysis software.

Application of this method when shrinkage data is available

The main factors affecting the accuracy of the prediction from the theoretical model described above are:

- Sensitive dependence of shrinkage on transition temperature and pvT data which, using the current measurement methods, cannot represent the behavior under actual injection molding conditions.

- No provision for determining molecular orientation and hence for unfilled materials the model does not predict anisotropy.
- Lack of provision for crystallinity effects.
- Lack of relaxation spectrum data for the viscoelastic calculation.

When a material has been shrinkage characterized by Autodesk Moldflow, the thermo-viscous-elastic model can be improved dramatically by accounting for the measured shrinkage results. This is done by using the theoretical model as one of the independent variables in a hybrid model that is correlated with measured shrinkage data in order to reduce the discrepancy between measured and predicted shrinkage. The resulting model is called a Corrected Residual In-Mold Stress (CRIMS) model.

The idea is illustrated below:

(a) Predicted Isotropic Residual Stress ⊠ **(b)** Error Correction, **(c)** Corrected anisotropic residual stress ⊠⊠ and ⊠y⊠, **(d)** Measured Shrinkage.

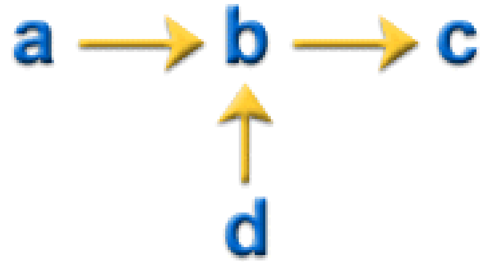


Figure 1: Corrected Residual In-Mold Stress (CRIMS) model

To illustrate the effect of this, consider the following graph.

(a) Shrinkage %, **(b)** Molding Condition Set Number, Measured Parallel, Corrected Parallel, Calculated the critical (isotropic).

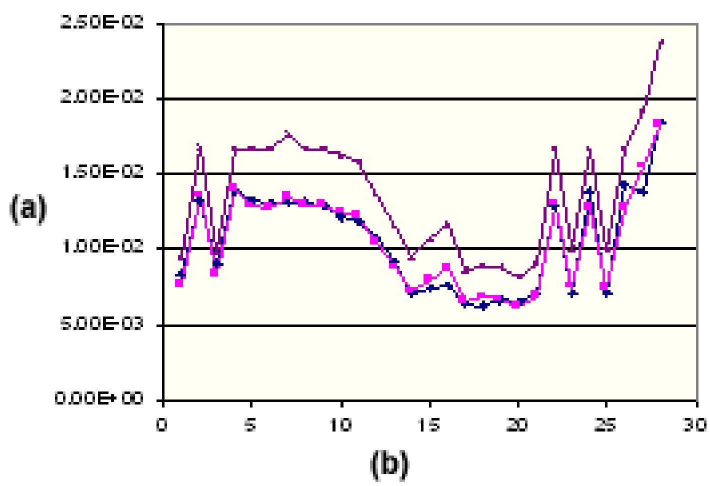


Figure 2: Parallel Shrinkage of Polypropylene

This graph shows the experimental shrinkage measured parallel to the flow direction for a polypropylene. Also shown are the theoretically calculated values of parallel shrinkage (using the thermo-viscous-elastic model) and the corrected value of parallel shrinkage. It is clear that the corrected values are in excellent agreement with the measured values. Similar improvement is noted in the perpendicular direction for the same polypropylene as shown below.

(a) Shrinkage %, **(b)** Molding Condition SetNumber, Measured Perpendicular, Corrected Perpendicular, Calculated the critical (isotropic).

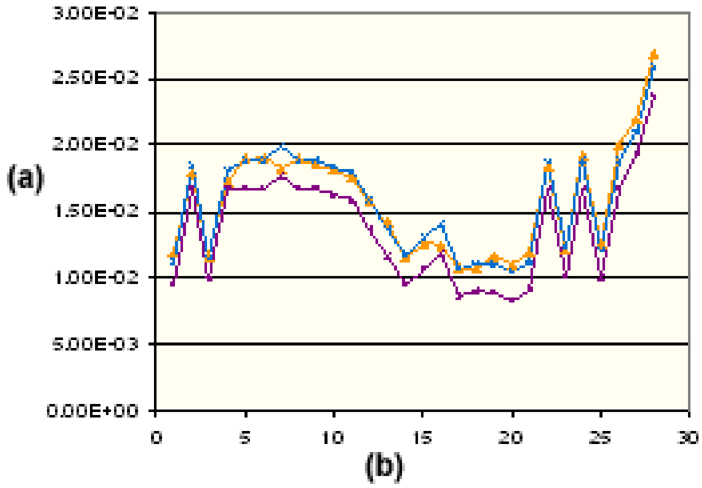


Figure 3: Perpendicular Shrinkage of Polypropylene

The correction concept also may be applied to fiber filled materials where it also gives excellent results. Below are some results on a PA66 that has 15% by weight of glass fiber reinforcement.

(a) Shrinkage %, **(b)** Molding Condition SetNumber, Measured Parallel, Corrected Parallel, Theoretical Parallel.

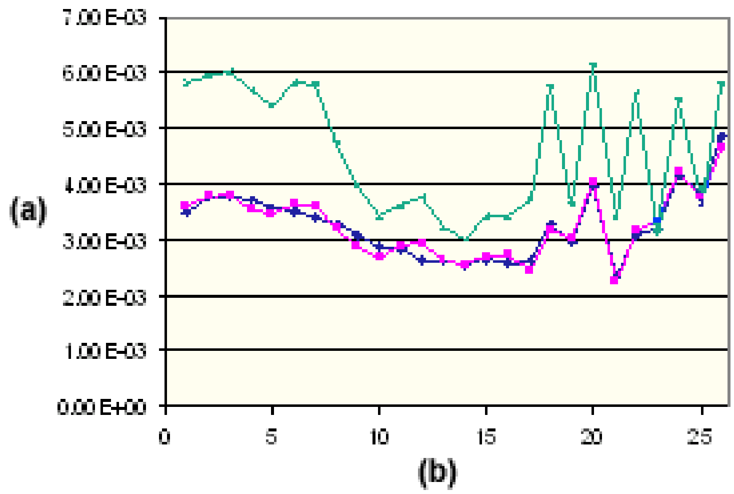


Figure 4: Parallel Shrinkage of PA66 15% GF

(a) Shrinkage %, **(b)** Molding Condition SetNumber, Measured Perpendicular, Theoretical Perpendicular, Corrected Perpendicular

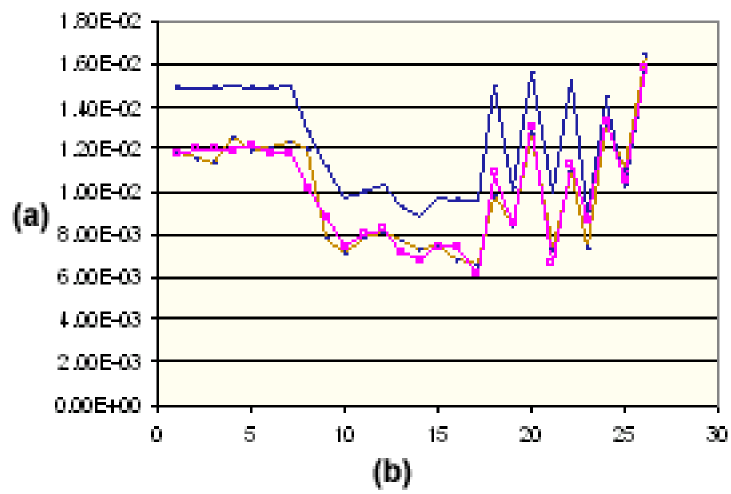


Figure 5: Perpendicular Shrinkage of PA66 15% GF

Application of this method when used with single variate analysis

Single variate analysis is a technique provided in the Autodesk Moldflow Warp analysis product to isolate the dominant cause of warpage and allow you to take targeted measures to reduce part warpage. It is discussed in detail in the Single Variate Analysis topics. Here we will consider how the residual stress method is applied in the single variate analysis context.

The Fill+Pack analysis of the filling and packing phases outputs the following information which serves as input for the residual stress calculations:

- Generalized forces data, i.e., the membrane forces (Nx,Ny,Nx⊠y) and bending moments (Mx,My,Mx⊠y), on each element in the local elemental axis system,
- Material orientation angles for each element,
- Mechanical properties for each element (the longitudinal Young's modulus, the transverse Young's modulus and the Poisson's ratio).

Single variate analysis is based on the concept that the causes of warpage fall into three categories:

- Differential cooling,
- Differential shrinkage,
- Orientation effects.

This concept is concerned with shrinkage rather than stresses. Therefore, to isolate the cause of warpage in terms of the above three effects when using the residual stress model, we need to calculate generalized strains from the given generalized forces, then decompose the strains into components due to the differential cooling, different shrinkage and orientation effects, and finally convert the modified strains back into corresponding generalized forces. Separate stress analyses are then performed to obtain warpage results for each effect. The relevant equations for the calculations are summarized below.

Hooke's law for a transversely isotropic material (the mechanical properties are isotropic in all planes perpendicular to the fiber axis , that is axis 1) can be expressed as:

⊠Sf⊠

where ⊠⊠⊠,⊠⊠,⊠⊠TSf=1E1⊠−⊠⊠E1⊠0−⊠⊠E1⊠1E2⊠0001G ,
⊠⊠⊠,⊠⊠,⊠⊠T , and the compliance matrix is given by

Alternatively, Hooke's law can be written in terms of a stiffness matrix as follows

⊠Df⊠

with

Df=11−⊠⊠⊠⊠⊠1⊠⊠⊠⊠1⊠0⊠⊠⊠1⊠E2⊠000G

Given the material orientation angle ⊠we transform the compliance matrix and the stiffness matrix from the oriented system to the local elemental system.

The generalized strains are given by

⊠⊠⊠⊠⊠=1h⊠1⊠S1⊠S1⊠S2⊠S2⊠S2⊠S3⊠S3⊠S3⊠⊠NxNyNx⊠
KxKyKx⊠=12h3⊠1⊠S1⊠S1⊠S2⊠S2⊠S2⊠S3⊠S3⊠S3⊠⊠MxMyMx⊠

where ⊠and ⊠are the strain and the curvature vectors respectively.

The following equations convert the strains and the curvatures back to the membrane forces and bending moments:

NxNyNx⊠=h⊠D1⊠D1⊠D1⊠D2⊠D2⊠D2⊠D3⊠D3⊠D3⊠⊠⊠⊠⊠⊠
MxMyMx⊠=h312⊠D1⊠D1⊠D1⊠D2⊠D2⊠D2⊠D3⊠D3⊠D3⊠⊠KxKyKx⊠

To isolate the effects of warpage, we replace ⊠ and ⊠ by the decomposed components, then recalculate the membrane forces and bending moments and use the new values in the structural analysis.

Residual stress shrinkage prediction method

Use this dialog to show the coefficients for a Corrected Residual In-Mold Stress (CRIMS) model.

CRIMS Model Coefficients dialog

This dialog shows the coefficients for the Moldflow proprietary Corrected Residual In-Mold Stress (CRIMS) shrinkage model, as obtained from shrinkage testing using the default Flow/Fiber data set.

NOTE: If the shrinkage model for the selected material is set to CRIMS, and the **Use CRIMS** option is set to the default (“change solver parameters to be consistent with the CRIMS model”), any changes that you make to Fiber orientation prediction solver parameters will be overwritten in the analysis. If you wish to use non-default settings for Fiber orientation prediction solver parameters, either change the **Use CRIMS** option setting, or select a different shrinkage model.

CRIMS Model Coefficients for RSC Fiber model dialog

This dialog shows the coefficients for the Moldflow proprietary Corrected Residual In-Mold Stress (CRIMS) shrinkage model, as obtained from shrinkage testing using the RSC Fiber model data set.

NOTE: If the shrinkage model for the selected material is set to CRIMS, and the **Use CRIMS** option is set to the default (“change solver parameters to be consistent with the CRIMS model”), any changes that you make to Fiber orientation prediction solver parameters will be overwritten in the analysis. If you wish to use non-default settings for Fiber orientation prediction solver parameters, either change the **Use CRIMS** option setting, or select a different shrinkage model.

Shrinkage prediction method for 3D models

For injection molded parts, the part is constrained in the mold. During the solidification of an injection molded part, shrinkage of the solidified layer is prevented by two mechanisms. Once the part is ejected from the mold, these residual stresses will be released in the form of shrinkage deformation

There are two mechanisms preventing shrinkage of the solidified layer while still in the mold. Firstly, adhesion to the mold walls restrain (at least the outer skin of) the solid layers from moving, and secondly, the newly formed solid surface will be kept fixed by the stretching forces of melt pressure.

In-cavity residual stresses build up during solidification. Due to the nature of constrained quenching, the residual stresses distribution is largely determined by the varying pressure history, coupled with the frozen layer growth. Once the part is ejected from the mold, these residual stresses will be released in the form of shrinkage deformation. If the initial strains, which are equivalent to the in-cavity residual stresses, are uniform, the part will shrink uniformly without any warpage and post-mold residual stresses. Warpage is caused by variations in shrinkage throughout the part.

Two types of shrinkage variations are considered:

Shrinkage variations from region to region (differential shrinkage effects)	For typical thin-walled parts, this form of shrinkage variation can be divided into variations in the thickness direction of molded parts, mainly caused by the differential cooling and variations from surface region to surface region.
Shrinkage variation in different directions (orientation effect)	The difference between parallel and perpendicular shrinkage, and anisotropic material properties relating to the fiber-orientation distribution are one of the main causes of part warpage for fiber-filled thermoplastics.

The shrinkage of injection molded parts depends on the thermodynamic behavior of the material during processing. For simplicity, we assume linear elastic behavior in the solidified part and purely viscous behavior in the melt.

It is reasonable to approximate the linear shrinkage using the formulation:

$$\varepsilon_i=\int_{T_0}^{T_r}\alpha_i(T)\,dT$$

where

- $\alpha_i(T)$ is the linear thermal expansion coefficient (CTE) at temperature T in the *i-th* principal direction.
- T_0 is the temperature when the local cavity pressure reached the atmospheric condition. This value is obtained from the flow simulation.
- T_r is the room temperature.

The 4-node first-order tetrahedral element is appropriate for 3D flow simulation. However, if the first-order tetrahedral element is used for the Warp analysis of typical thin-walled parts or thin-walled areas of complex three-dimensional parts, the notorious shear locking problem will make the structural response very stiff [1]. Shear locking, or parasitic shear, is caused by an inaccuracy in the linear displacement field of a linear tetrahedral element. It can be exacerbated by elements with large aspect ratios. On the other hand, the high aspect ratio tetrahedral elements may not be avoidable if the computational cost is to be kept low. Therefore, the first-order tetrahedral element is not suitable for the thin-walled areas of injection molded parts.

A hybrid element scheme has been designed for the 3D Warp analysis. 4-node first-order tetrahedral elements are used in the 3D solid areas, and 10-node second-order tetrahedral elements are used in the thin-walled areas. Transitional 5-9 node tetrahedral elements are used in the transitional areas which connect the thin-walled and thick areas.

For fiber-filled polymers, the flow-induced fiber orientation is also simulated. Then the moduli, Poisson's ratios and thermal expansion coefficients of the composite material can be determined using the calculated fiber orientation and mechanical properties of polymer matrix and the fiber [2, 3, 4]. This information is put into structural tetrahedral elements, in which the 3D orthotropic stress-strain relationship is used.

[illegible]

[illegible]1–[illegible]–[illegible]–[illegible]–2[illegible]Ex[illegible]y[illegible]z

3D warpage simulation normally requires significant computational time particularly if the number of elements is very large and if there is a big thin-walled region. An efficient preconditioned conjugate gradient iterative solver is implemented for reducing the memory requirement and computational time.

References:

[1] K.J. Bathe, *Finite Element Procedures*, Prentice Hall Inc.(1996).

[2] S.G.Advani and C.L.Tucker III, *The Use of Tensors to Describe and Predict Fiber Orientation in Short Fiber Composites*, J. Rheol., 31,751-784(1987).

[3] G.P.Tandon and G.J.Weng, *The Effect of Aspect Ratio of Inclusions on the Elastic Properties of Unidirectional Aligned Composites*, Polym. Compos. 5(4),327-333(1984).

[4] R.A.Schapery, *Thermal Expansion Coefficients of Composite Materials Based on Energy Principles*. J. Compos. Mater., 2, No.3, 380-404, (1968).