Autodesk® Moldflow® Insight 2012

AMI Warp Analysis

Autodesk®

Revision 1, 22 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

1

A Warp analysis is used to diagnose the cause of warping and recommend a solution, such as gate location changes, design parameter changes, or reduction of wall thickness variations.

Warp analysis is supported for Midplane,Dual Domain and 3D analysis technologies.

Warpage is affected greatly by fiber orientation. Warpage in an injection molded part is caused by variations in shrinkage that occur:

- From region to region in the part.
- Through the thickness of the part.
- Parallel and perpendicular to the direction of material orientation.

Any attempt to predict the likelihood and amount of warpage for a particular component must first account for these shrinkage variations.

The material that is used influences the warpage of the part. Sometimes you must choose a material with low, uniform shrinkage to achieve dimensional accuracy. These materials are usually more expensive, but you may be able to use a cheaper material and save costs by observing shrinkage and warpage design principles.

NOTE: Differential shrinkage caused by fiber orientation is one of the main causes of warpage.

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.

Warp analysis

A Warp analysis is used to determine whether a part molded with a thermoplastic material will warp.

Setting up a Warp analysis

The following table summarizes the setup tasks required to prepare a Warp analysis.

The setup tasks below are for non fiber-filled, or fiber-filled thermoplastic materials.

Setup task	Analysis technology
Meshing the model	
Mesh orientation diagnostic	
Checking the mesh before analysis	
Selecting the molding process	
Analysis sequence	
Selecting a material	
Process settings	

Optional setup tasks

Setup task	Analysis technology
Optional tasks on page 6	

Warp analysis

Use this dialog to specify settings for a Warp analysis.

Process Settings Wizard dialog—Warp Settings

This page of the **Process Settings Wizard**, which can be accessed by clicking (**Home tab > Molding Process Setup panel > Process Settings**), is used to specify the Warp analysis related process settings for the analysis sequence.

NOTE: Some of the items listed below may not be available on the current dialog. This is dependent on the analysis technology, molding process and analysis sequence selected.

Warpage analysis type	Allows you to select the type of Warp analysis to run.
Upgrade tetrahedral elements to second order	Specify whether the 4-noded tetrahedral elements (first-order elements) created by the mesher should be upgraded to 10-noded tetrahedral elements (second-order elements) in the 3D Warp analysis.

Stress result(s) to output	Specify which laminate-based stress-related results the solver will output.
Stress analysis type	This option is used to select the type of Stress analysis to run to predict paddle shift/wire sweep.
Consider gate surface and cold runners?	Specifies whether cold runners and/or gate surface elements, if present in your model, are taken into consideration during Warp or Stress analysis.
Consider corner effects	Select this option if you want the Warp analysis to calculate and account for deformations due to mold-restraint induced differential shrinkage.
Matrix solver	Select the equation solver to be used in the Warp analysis.
Consider mold thermal expansion	Select this check box if you want Warp or Stress analysis to consider the effect of mold thermal expansion on the warpage and/or molded-in stress levels in the part.
Include cold runners	By default tetrahedral cold runners are exlcuded by warpage calculations.
Use mesh aggregation and 2nd-order tetrahedral elements/mesh options	Controls whether 3D Warp analysis should employ mesh aggregation.
Use GPU card if available	Specifies whether you want numerical calculations to be performed on a GPU (Graphics Processing Unit) card when running an analysis.
Number of threads for parallelization	Specify the number of threads to be used for parallel solution.
Use AMG matrix solver	Select whether the Algebraic Multigrid (AMG) matrix solver for 3D warp is to be used.

Warp analysis types

2

There are different warp analysis types that can be selected.

The following Warp analysis types are available and can be selected from the Process Settings Wizard:

Buckling Determines whether the warpage of the part is stable or unstable.

If the buckling analysis indicates that the warpage is stable (critical load factor > 1), the deflection results obtained from the buckling analysis provide a good indication of the final deformed shape of the part.

If the buckling analysis indicates that the warpage is unstable (critical load factor < 1), you will need to run a large deflection analysis to determine the final deformed shape of the part.

This analysis is performed on Midplane meshed models.

Small deflection Select this analysis type if you expect the warpage of the part to be stable. The small deflection analysis provides the final deformed shape of the part, assuming linear stress-strain behavior within the part.

This analysis can be performed on Midplane, Dual Domain and 3D meshed models.

Large deflection Select this analysis type if you expect the warpage of the part to be unstable, as determined from a previous automatic or buckling analysis, or the warpage of the part is borderline stable/unstable and/or you want the most accurate prediction of the shape of the part. The large deflection analysis provides the final deformed shape of the part, allowing for nonlinear stress-strain behavior within the part.

This analysis can be performed on Midplane and 3D meshed models.

Automatic Select this option if you want the solver to identify whether the warpage is stable or unstable. If the warpage is determined to be unstable and you are using a Midplane mesh, this option will automatically run an additional large deflection analysis to determine the true final deformed shape of the part. Depending on your mesh type, automatic analysis first runs a buckling analysis, a large deflection analysis is run if the Eigenvalue is less than 1.5, otherwise a small deflection analysis is performed.

Preparing the model for Warp analysis 3

The Warp analysis uses the results of the Cool analysis and the Fill analysis to predict shrinkage and warpage of the part. Modeling tasks and an analysis sequence must be set up before a Warp analysis is run.

Modeling tasks

Before running a Warp analysis, you should ensure that the correct processing conditions have been set, in particular, Packing Time and Packing Pressure, and that the gate and feed systems have been modeled correctly.

- The model is meshed.
- The mesh is oriented: **Mesh tab > Mesh Repair panel > Orient Elements**.
- The correct processsing conditions have been set, in particular Packing Time and Packing Pressure.
- The gate and feed system has been modeled correctly.

Analysis setup

The recommended analysis sequence to run for a Warp analysis is “Cool+Fill+Pack+Warp”.

The Cool analysis ensures that the effect of the cooling system on warpage can be taken into account. Here you can either specify a cooling time or select automatic cooling time calculation.

It is important to run a Cool analysis before the Warp analysis so that the information gained from the Cool analysis can be taken into account. If you do not run a Cool analysis first, differential cooling effects, such as temperature differences from one side of the mold to the other, will not be included in the Warp analysis calculations. You can specify a cooling time or select an automatic cooling time calculation.

Optional tasks

4

There are optional set-up tasks that you can perform for a Warp analysis.

Constraining the model for a Warp analysis

For a Warp analysis, constraints are applied to the model nodes to prevent rigid body motion (global translations and rotations) of the model, in response to the natural warpage of the part.

NOTE: General rigid-body motion in space involves six components (three orthogonal translations and three orthogonal rotations). This means that the minimum number of constrained degrees of freedom that must be set in the model is also six. In practice, you must decide whether the global coordinate system or a local coordinate system best simulates your perception of the physical situation being modeled.

Automatic vs Manual constraints

By default, and for most cases, automatic constraints can be used to predict the typical deformation of a part.

If no manual constraints were set, then constraints are set automatically, and the deflection of the part is measured using the *best-fit* technique.

Although an automatic warpage calculation is often sufficient, it can also be useful to set manual constraints on a part in order to predict how it will warp if it is used in particular conditions, or if it is fixed to another object in an assembly for example.

You may want to set non-zero displacement constraints on specific nodes in order to consider assembly-induced deformations and stresses. In order to do this, you need to set rigid body motion constraints on the model first.

Figures 1 and 2 below show how warpage simulations can vary depending on the type of constraints that were set on the part.

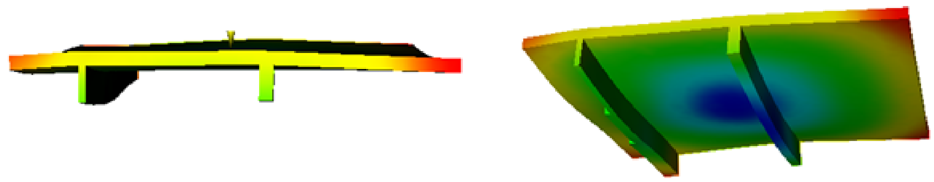


Figure 1: Typical deformation of a part with automatic constraints (scale factor=10.00)

However, setting specific constraints on a part enables early prediction of any unwanted behavior of a part in real conditions of use.

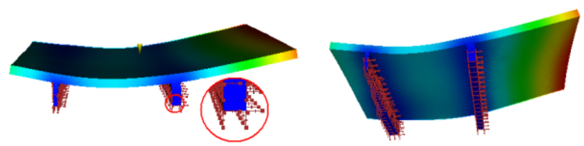


Figure 2: Simulation of the constraints that will be applied on the part when in use, and deflection resulting from these constraints. (scale factor=10.00)

Constraining the model for a Warp analysis

You can use specific constraints instead of automatic constraints during a Warp analysis. Setting Warp constraints is useful in order to predict how a part will warp if it is used in particular conditions, or if it is fixed to another object in an assembly.

NOTE: If some of the constraints set on the part already fix the rigid body motion, then you don't have to set extra constraints.

Settings Warp constraints

You must have a Midplane, Dual Domain or 3D meshed model, and use an analysis sequence including a Warp analysis.

To perform a Warp analysis, you need first to prevent rigid body motion (free rotation in space) of the part by constraining 6 degrees of freedom (translational and/or rotational) for each cavity. Once the rigid body motion is fixed, you can set the displacement constraints to simulate real conditions of use for your part.

1. If the model nodes are not visible, turn on the appropriate layer in the **Layers** pane.
2. If you want to use a local coordinate system when setting constraints, first define and activate the local coordinate system.
3. Click **Boundary Conditions tab > Constraints and Loads panel > General Constraint**.

The **General Constraint** dialog is displayed.

4 Fix the rigid body motion by defining a reference plane with 3 nodes as follows:

   a Use the cross-hairs to select the first node of the reference plane, then pin this node by setting **Fixed** constraints on the **X**, **Y**, and **Z** translations.

   b Select **Warp Analysis** from the **Use constraint in** drop-down menu, and then click **Apply**.

   This defines the origin of the reference plane.

   c Select a second node, then set **Fixed** constraints on the **Y** and **Z** translations.

   d Make sure the **Warp Analysis** remains selected in the **Use constraint in** drop-down menu, and then click **Apply**.

   e Select a third node, then set **Fixed** constraints on the **Z** translation.

   f Make sure the **Warp Analysis** remains selected in the **Use constraint in** drop-down menu, and then click **Apply**.

   The rigid body motion constraints are now fixed.

5 Set the displacement constraints.

NOTE:

When displacement constraints are set, the undeformed part is used as a reference in order to define the final shape.

During the warpage analysis, the displacement constraints are adjusted to fit the desired final shape of the part.

---

   a Use the cross-hairs to select the node(s) for which you want to set the constraint.

   b Specify for each of the X, Y, and Z directions whether the translational or rotational degree of freedom at the node(s) should be Free, Fixed, or set to a specified distance or angle.

   c Select **Warp Analysis** from the **Use constraint in** drop-down menu.

   d Click **Apply** in the **Set Constraints** dialog or right-click in the model pane and select **Apply.**

6 Once you have finished applying constraints, right-click in the model pane and select **Finish Constraints**.

Excluding elements from warp calculation

You can exclude any type of element from the warp calculation.

Excluding elements from the warp calculation can be useful in the following cases:

- If you want to isolate a cause of warpage in a part by excluding some structural areas like ribs or part inserts for example.

- If you want to reduce the analysis time by excluding entire parts when in multi-cavity mode.

This option is available for Midplane and 3D studies.

Excluding elements from warp calculation

You can exclude any type of element from the warpage calculation.

Excluding elements from warp calculation

Excluding elements for the warpage calculation is particularly useful when trying to isolate a cause of warpage in a part, or simply to reduce the analysis time.

You need to have meshed elements in a single part or multi cavity model.

1. Select the element, group of elements or part that you want to exclude from the analysis.
2. Right-click, and select **Properties**.
   The **Part surface** dialog opens.
3. Select the **Exclude from warpage calculation** checkbox.
4. Click **OK**.
5. Double-click **Start Analysis!** from the **Study Tasks** pane, or (**Home tab > Analysis panel > Start Analysis**) to start the analysis.

Mesh aggregation for 3D Warp analysis

5

Mesh aggregation is implemented as an option in the process settings for 3D Warp analysis to improve solution speed and reduce memory requirement for 3D analysis of typical thin-walled plastic parts. This option is enabled by default.

Mesh aggregation is a technique which reduces the number of layers of elements in a tetrahedral mesh to two, while upgrading the elements to 2nd-order, 10-node tetrahedral elements. Because the 2nd-order element is a high-quality element, even two layers of 2nd-order tetrahedral elements can provide equally good deflection (warpage) results for thin-walled parts compared to six layers of 2nd-order elements. By contrast, a tetrahedral mesh used for 3D Fill+Pack analysis typically has six or more layers of 1st-order, 4-node tetrahedral elements.

When mesh aggregation is enabled, two different meshes are used for 3D Fill+Pack and 3D Warp analyses, respectively. Using mesh aggregation, there are fewer elements and fewer unknowns in the 2-layer mesh, and therefore, the 3D Warp memory requirement is reduced significantly, and the element-level computations and equation solution time are much less.

If the part is solid or thick (true 3D geometry), using mesh aggregation is not recommended. Turning off the mesh aggregation option causes the original mesh to be used for the analysis. This increases analysis time and memory requirement but improves accuracy for true 3D parts.

When the mesh aggregation option is disabled, you can access additional mesh control options. For example, you can specify whether the 1st-order tetrahedral elements created by the mesher should be upgraded to 2nd-order tetrahedral elements for the 3D Warp analysis (without reducing the number of layers of elements). In thin-walled areas of the part, using 2nd-order elements will improve the accuracy of the prediction. The **Upgrade tetrahedral elements to second order** option is set to **Automatic** by default. In this case, 3D Warp automatically uses a scheme in which 1st-order, 4-node tetrahedral elements are used in the true 3D, solid areas, and 2nd-order, 10-node tetrahedral elements are used in the thin-walled areas, with transitional elements connecting these areas.

NOTE: Mesh aggregation is not available for 3D Gas-Assisted injection molding.

Mesh aggregation for 3D Warp analysis

To access the mesh aggregation option, ensure that you have specified an analysis sequence that includes Warp.

Mesh aggregation for 3D Warp analysis

The **Use mesh aggregation and 2nd-order tetrahedral elements** option is selected by default.

1. Open a 3D model which requires Warp analysis, and make sure an analysis sequence which includes Warp is set.
2. Click **Home tab > Molding Process Setup panel > Process Settings**, or double-click the Process Settings icon in the **Study Tasks** pane. Click **Next** one or more times until the Warp Settings page appears.
3. Clear the **Use mesh aggregation and 2nd-order tetrahedral elements** checkbox, to disable the mesh aggregation option.

   When the mesh aggregation option is disabled, you can access additional mesh control options.
4. Click **Advanced options** to access additional mesh control options.
5. Click **OK** to accept the changes, or **Cancel** to exit without any changes.
6. Click **Finish**.

Mesh aggregation for 3D Warp analysis

Use this dialog to specify settings for a 3D Warp analysis.

Warp Settings dialog—3D Warp advanced options

The **3D Warp advanced options** dialog allows you to specify analysis options for a 3D Warp analysis if you have not chosen to enable mesh aggregation. Options include:

Small deflection warpage analysis	This type of analysis allows you to enable the **Isolate cause of warpage** option if you want the small deflection that you are setting up to output information about the dominant cause of warpage.
Large deflection warpage analysis	Select this analysis and, optionally, specify the settings for the **solver parameters** to run a large deflection Warp analysis.
Upgrade tetrahedral elements to second order	This option is used to specify whether the 1st-order, 4-node tetrahedral elements created by the mesher should be upgraded to 2nd-order, 10-node tetrahedral elements in the 3D Warp. In thin-walled areas of the part, 2nd-order elements will improve the accuracy of the prediction.

NOTE: To access this dialog, deselect the **Use mesh aggregation and 2nd-order tetrahedral elements** check box in the **Warp Settings** page of **Process Settings Wizard**, and click the **Advanced options** button.

Parallel solution method for 3D Warp analysis

6

Parallel solution technology is implemented as an option in the process settings for 3D Warp analysis to improve solution speed, especially for large models.

The average expected solution time speed-up factor for 2-CPU systems is 1.3 and for 4-CPU systems is 1.5, where the speed-up factor is calculated as the ratio of serial solution time to parallel solution time (based on elapsed wall clock time).

The parallel solution method is supported for shared memory multi-processor (SMP) systems, also known as multiple core systems. In SMP systems, all physical processors (cores) are in the same computer and access the full system memory, so data sharing is fast.

NOTE: Distributed memory clusters are not supported.

If you have hyperthreading enabled, then the number of processors available will appear to be twice the number of physical processors. However, this does not result in the most efficient parallel execution. For best results, the number of threads specified for parallelization should not exceed the number of physical processors available on the system.

NOTE: For parallel analysis, the AMG matrix solver will always be used. The option to disable the AMG solver is only available for single-threaded analysis.

Parallel solution method for 3D Warp analysis

To access the parallel solution option, ensure that you have specified an analysis sequence that includes Warp.

Enabling the parallel solution

The **Number of threads for parallelization** option is set to **Single thread (No parallelization)** by default. This means the parallel solution method will not be used. You can specify the number of threads to be used for parallel solution.

1 Click **Home tab > Molding Process Setup panel > Process Settings**. Click **Next** one or more times until the Warp Settings page appears.
2 Change the **Number of threads for paralellization** from **Single thread (no parallelization)**, to your preference.

If you set the **Number of threads for parallelization** option to **Maximum**, the analysis will be run using the maximum number of physical processors available for parallelization. This includes multiple cores, but does not include additional logical processors made available by enabling hyperthreading.

If you set the **Number of threads for parallelization** option to **Specify number of threads**, you can specify the number of threads you want to be used for parallelization.

NOTE: The number of threads specified should not exceed the number of physical processors available.

---

3 Click **Finish**.

Contributors to warpage

7

There are several factors that could cause a part to distort.

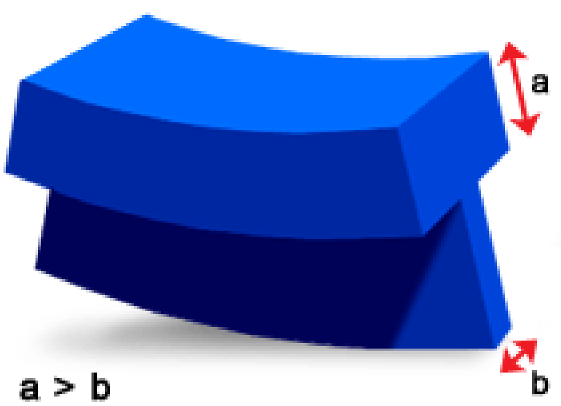
When considering the contributors to warpage, it is convenient to identify shrinkage due to:

- Variation in shrinkage from region to region (differential shrinkage).
- Temperature differences from one side of the mold to the other (differential cooling).
- Variations in the magnitude of shrinkage in directions parallel and perpendicular to the material orientation direction (orientation effects).

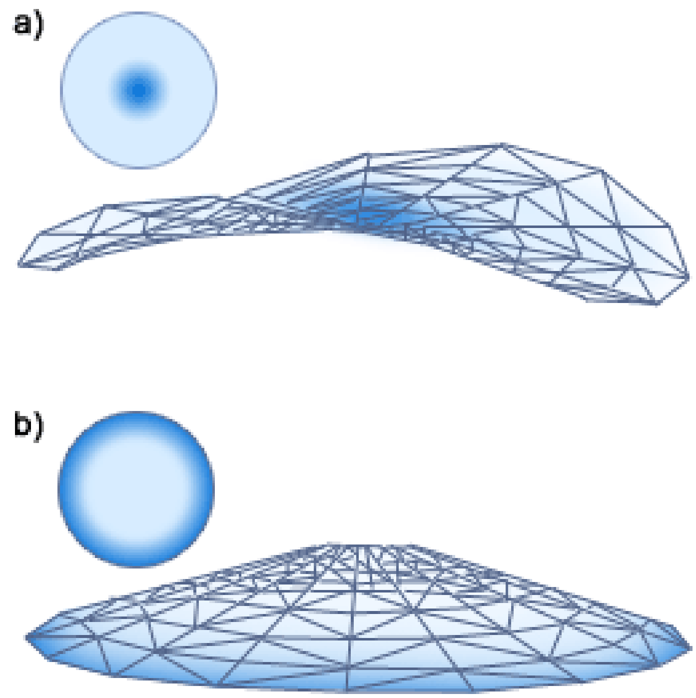
Each of these types of shrinkage will contribute to the total warpage of the product.

Differential Shrinkage

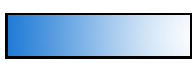
This type of shrinkage is often caused by variation in crystalline content and volumetric shrinkage. The following figure shows a thin rib attached to a thick top. In general, the cooling rate of the top will be lower than that of the thin section. The top will have increased crystalline content and therefore, will shrink more and cause the warpage shown.



The following figure (a) shows saddle warping of a centrally gated disk with high shrinkage around the gate. Conversely, if the shrinkage is higher around the outer part of the disk, the resulting warpage may cause the disk to dome, shown in (b).



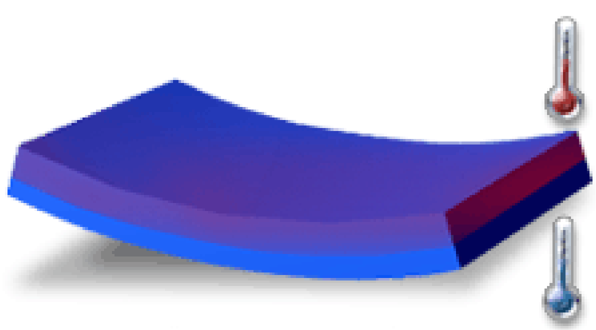
High Shrinkage



Low Shrinkage

Differential Cooling

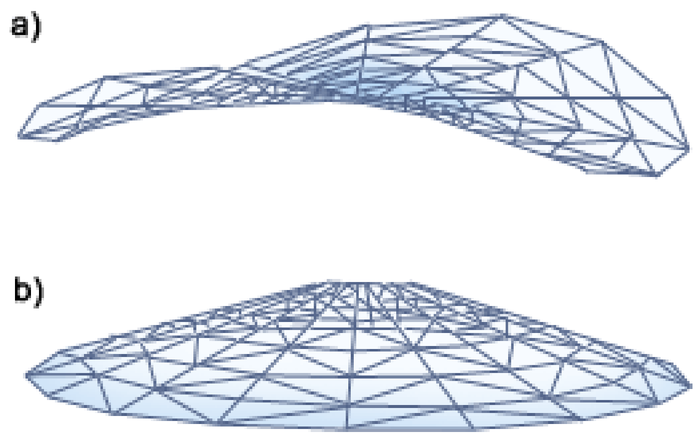
Shrinkage due to differential temperature typically results in bowing of the component, as shown in the figure below. Usually this type of shrinkage is due to poor cooling system design.



While the part is in the mold, temperature differences from one side of the mold to the other cause variations in shrinkage through the thickness of the component. In addition to this, any temperature differences at ejection will cause further warpage as both sides of the part cool to room temperature.

Orientation Effects

Orientation causes variation in the magnitude of shrinkage in directions parallel and perpendicular to the material orientation direction. This type of shrinkage can produce warpage similar to that of differential shrinkage. Figure (a) below shows the warpage when parallel shrinkage is greater than perpendicular shrinkage. On the other hand doming can be produced if the perpendicular shrinkage is higher than parallel shrinkage, see (b).



Warpage due to corner effects 8

Warpage is a very common problem in parts that have corners.

There are two main causes for the warpage in corners:

Heat build-up.	The ability to extract heat from corner areas in the part is lower which results in asymmetric cooling and thermally induced stresses.
Differential shrinkage	Due to mold restraint, the shrinkage in the thickness direction is much greater than in-plane shrinkage in corner areas of the part, which results in further deformation.

The thermal effect is taken into account by default in all Autodesk Moldflow Insight cooling and warpage simulations. The mold-restraint induced differential shrinkage contribution is taken into account if the **Consider corner effects** option is turned on in the advanced options for the Warp analysis.

For further information about the differential shrinkage effect in corners, and the mathematical approach to calculate and account for this effect, please refer to the following published paper:

Ammar, A., Leo, V., and Regnier, G., “Corner deformation induced by shrinkage anisotropy of injected thermoplastics: Experimental study and numerical approach”, PPS-17 (2000)

9 Single variate analysis method

Single variate analysis is a method employed in the Warp analysis product to isolate the cause of warpage by regarding the differences in shrinkage values within the part, the fundamental cause of warpage, as arising from three independent contributors to shrinkage variations.

The total shrinkage in an element is defined to be:

$$S = Sd_c + Sd_s + So_e$$

where Sd_c , Sd_s and So_e are the components of shrinkage due to differential cooling, differential shrinkage and orientation, respectively.

Single variate analysis method—stable warpage

10

If the part warpage is stable, that is, the deflections of the part can be adequately predicted by small deflection analysis assuming linear stress-strain behavior, the contribution of each of the three shrinkage variates to the total deflection can be assumed to be linear and additive.

To determine the part warpage attributable to each of these variates, three individual analyses are run where in each case the contribution of just one contributor is taken into account by eliminating the effect of the other two. Visual inspection of the magnitude of the part deflections in each single variate case indicates the dominant cause of warpage. This functionality is available for Midplane, Dual Domain and 3D analysis technologies.

Differential Cooling Single Variate Analysis

This analysis calculates the shrinkage contribution due solely to differential cooling.

For the top and bottom of each element, and for the perpendicular and parallel directions, the differential cooling component of the shrinkage is determined by subtracting the differential shrinkage and orientation effects contributions from the total shrinkage determined from the shrinkage calculations as part of the Fill+Pack analysis.

For example, for the parallel orientation direction of the top of the element, the contribution from cooling would be,

S¯⊠¯⊠T–Slin–S¯⊠Slin

where

- S⊠s the parallel shrinkage value from the packing analysis.
- ⊠T is the difference between the temperature on the top of the element and room temperature.
- ⊠¯ is the average of ⊠T and ⊠B
- Slin is the linear shrinkage value, as defined for the differential shrinkage variate.

Differential Shrinkage Single Variate Analysis

This analysis calculates the shrinkage contribution due solely to differential shrinkage.

The shrinkage contribution due to orientation effects is eliminated by assigning to each element the same shrinkage value in both the parallel and perpendicular material orientation directions. This value is the linear shrinkage value Slin, given by:

Slin=1–1–Siso

where Si⊠⊠ is the isotropic shrinkage determined in the shrinkage calculations as part of the Fill+Pack analysis.

The contribution of differential cooling is eliminated by assigning this Slin value to both the top and the bottom of the element.

Orientation Effects Single Variate Analysis

This analysis calculates the shrinkage contribution due solely to orientation effects.

The shrinkage contribution due to differential shrinkage is eliminated by subtracting the linear shrinkage value Slin, from the parallel and perpendicular shrinkage results determined in the shrinkage calculations as part of the Fill+Pack analysis.

The shrinkage contribution due to differential cooling is eliminated by assigning the same parallel and perpendicular shrinkage value to both the top and the bottom of the element.

Single variate analysis method—unstable warpage

11

Unstable warpage leads to buckling of components. The contribution of each of the three shrinkage variates to the total deflection can no longer be assumed to be linear.

In this case, a so-called **sensitivity analysis** is used. This functionality is available for Midplane analysis technology only.

The aim of the sensitivity analysis is to determine the change in load factor for a known change in the shrinkage. The load factor indicates at what factor of the actual applied loads, in the case of Shrink analysis these being the loads internal to the part, the part warpage becomes unstable, leading to buckling. A load factor less than one indicates the actual loads are sufficient to result in buckling of the part. A load factor greater than one indicates that the warpage is stable as the onset of buckling has been determined to be at a load magnitude higher than the actual loading.

Assuming that the shrinkage components can be varied independently, the load factor, □ can be regarded as a function of these components, that is,

□□□d□,Sd□,So□

To solve a warpage problem, we wish to know which component to change to increase the load factor, that is to make the warpage stable. One way of doing this is to take the partial derivatives of □with respect to each component. Unfortunately this cannot be done analytically as there is no known function relating □to the components. Instead the partial derivatives are approximated.

Suppose that the load factor from a buckling analysis using the total shrinkage is □ . Now increase one of the shrinkage components by some amount, □say, to give a new total shrinkage S□For example, if we increase the differential shrinkage component, we would have:

S□□Sd□+1+□d□+So□

If this value of S□s used in a buckling analysis to obtain a new load factor □□, the derivative of □with respect to the differential shrinkage component is approximated as follows:

∂□Sd□≈□□□□□d□

See *Figure 3: Load factor as a function of shrinkage* on page 22

(a) Load Factor, **(b)** Shrinkage

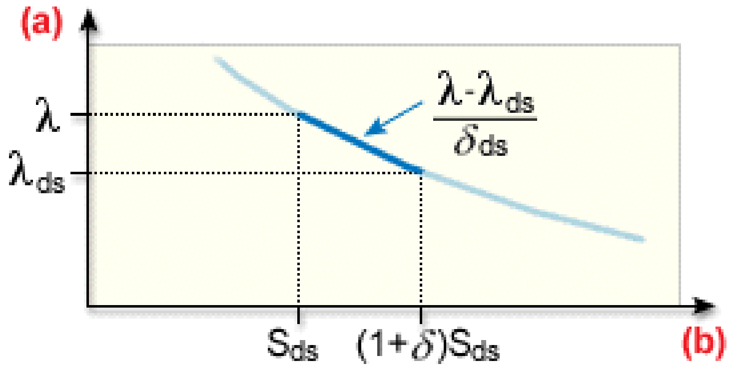


Figure 3: Load factor as a function of shrinkage

The above holds for one element only. For a real part, each element has a value of δ_{ds}. To cope with this, the elemental changes are combined into a single measure of shrinkage change. The norm of the changes are used in the elemental shrinkages to obtain the single value.

This norm is defined as follows:

$$\|\delta\| = \sum_{i=1}^{N} \delta_{ds_i}^2$$

where *N* is the number of elements in the model.

The value $(1 + \delta)$ is called the sensitivity factor which is a program input.

Similarly, the sensitivities of the load factors for differential cooling and orientation effects variates, respectively, are calculated.

12 Reducing warpage due to differential cooling

The two main ways of influencing differential cooling are to change the cooling channel layout, or use mold inserts.

Perhaps one of the easiest things to alter is the temperature of the coolant. It may be useful to run two additional Cool analyses with the coolant inlet temperatures at say plus and minus 5°C with respect to the original inlet temperature used. The results from the Cool analyses can then be used in a single variant Cool analysis to give you some idea of the sensitivity of the part to variations in coolant temperature.

If it is not sufficient to simply alter the coolant temperature, you can consider the addition of extra cooling channels in troublesome regions or the use of mold inserts to reduce variations in cooling rates across the part.

Reducing warpage due to differential shrinkage

13

There are several ways to alter differential shrinkage.

The main ways of influencing differential shrinkage effects are:

- Designing packing profiles.
- Reducing part thickness variations.
- Using mold inserts.

NOTE: If you have previously reduced orientation effects, the differential shrinkage effects may be quite different to those found in the original part model because of changes to gate locations, or part thickness etc. Thus you must repeat the full Fill+Pack, Cool, Packing and Warpage analyses for the part model.

Use a packing profile

The first option to consider when reducing differential shrinkage is the use of a packing profile—this of course is dependent on the machine response time and its effectiveness may be limited for thin parts or parts with complex geometries.

NOTE: The advantage of using a packing profile to reduce warpage is that this does not involve changing the design specifications of the part.

If you decide to use a packing profile to reduce the differential shrinkage in a part, you must repeat the full Fill+Pack, Cool, Packing and Warpage analyses for the part model with reduced orientation levels.

Reduce part thickness variations

If you decide that changes to the wall thickness may be of more use in reducing differential shrinkage effects for the part, then you can proceed to alter the thickness in the region in question and re-analyze the modified part model. This can be an iterative process, until the differential shrinkage level is acceptable.

Use mold inserts

The final alternative for reducing differential shrinkage is to consider the use of mold inserts to reduce shrinkage due to variations in cooling rates. Again the process is to modify the part and re-analyze it.

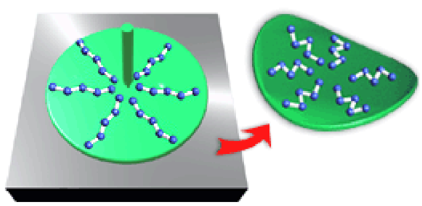
Reducing warpage due to orientation effects 14

There are several ways to alter orientation effects.

The three main ways to influence orientation (apart from choice of material) are to change:

- Molding conditions.
- Model thickness.
- Gate locations.

Orientation is caused by the combined effects of material shearing and freezing.



Change molding conditions

It may be possible to reduce orientation by changing the molding conditions (mold temperature, melt temperature, injection speed, etc). In contrast to the other two items above, this remedy does not require changes to the model or the mold so is the least expensive of the options to try.

Change gate locations

If molding conditions cannot be used to reduce orientation effects sufficiently, you must decide whether to change the gate type or location, or alter the model thickness. (Note that changing a gate location will not alter the design specifications of the part and may be an easier option to try on models with complex geometry and thickness variations.). Other changes to the gate, apart from simply changing its location, may include using an end gating, fan gate (only available in Autodesk Moldflow Insight) or multiple gates. All of these may be done without significantly altering the geometry of simple parts (assuming the mold has not already been cut!). Once you have decided on an alternative gate location (or type) you can re-analyze the modified model. This can be an iterative process, until the orientation level is acceptable.

Change model thickness

If you decide changes to the wall thickness will better reduce orientation effects for the model, then proceed to alter the thickness in the region in question and re-analyze the modified model. This too can be an iterative process, until the orientation level is acceptable.