Autodesk® Moldflow® Insight 2012

AMI Fill Analysis

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

Fill analysis

1

The Fill analysis predicts the thermoplastic polymer flow inside the mold in the filling phase. This analysis is often run as the first part of a Fill+Pack analysis sequence.

A Fill analysis calculates the flow front that grows through the part incrementally from the injection location, and continues until the velocity/pressure switch-over point has been reached. A Runner Balance analysis can be used with a Fill analysis to ensure equal pressure is delivered at each cavity.

A Fill analysis accepts a minimum of 8 and a maximum of 20 laminates across the part thickness to calculate results. The default number of laminates across the thickness is 12. You can change this value by editing the solver parameters.

If symmetrical thermal boundary conditions exist (Cool analysis results are not available), the analysis is based on a half-gap calculation and the actual number of laminates used in the calculation is half of the specified number, reflected symmetrically across the part. Running a Cool analysis prior to running a Fill analysis provides asymmetrical cooling information, and the full specified number of laminates is used to calculate results.

Before running a Fill analysis, pay close attention to the edge length of the mesh around high curvature areas on your model, and make sure the mesh in these areas is not too coarse. It is recommended that you mesh with a smaller edge length so the mesh approximates the corners correctly.

NOTE: Overmolding analyses and Core shift prediction use the Fill+Pack analysis sequence.

Supported analysis technologies

Select the most suitable analysis technology by using the following information:

Midplane analysis technology, Fill analysis	This predicts material behavior during the filling phase by analyzing a model which has a Midplane mesh. The mesh consists of 3-node, triangular elements. In general, the more elements there are in the mesh the more detailed the results. However, with more nodes the analysis time is also increased. This analysis is most accurate for thin-walled parts.
Dual Domain analysis technology, Fill analysis	This works by simulating the flow of the melt on both the top and bottom surfaces of the mold cavity (surface shell). This analysis is most accurate for thin-walled parts. The model has a surface shell mesh, consisting of 3-node, triangular elements. Consistency between the results on the opposite sides is maintained by using "connectors"—elements

with zero flow and heat resistance. The connectors are inserted automatically at locations determined according to the geometrical features of the model.

3D analysis technology, Fill analysis

This is a high-end flow simulation tool that helps you understand how plastic flows in thick-walled parts, or parts containing thick sections.

3D analysis technology allows the Fill analysis to directly analyze a volume mesh (4-node, tetrahedral mesh) of a solid model, removing the need for time consuming Midplane mesh preparation of a 3D part model.

Fill analysis

The Fill analysis is used to predict the thermoplastic polymer flow inside the mold in the filling phase.

Setting up a Fill analysis

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

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

You can use a Fill analysis in analysis sequences, such as Fill + Pack.

Setup task	Analysis technology
Molding processes	
Meshing the model	
Mesh orientation	
Checking the mesh before analysis	
Analysis sequence	
Selecting a material [1]	
Injection locations	
Velocity/pressure switch-over point	
Process settings [2]	

Optional setup tasks

1 Select a fiber-filled material to run a Fiber orientation analysis.

2 Ensure the **Fiber orientation analysis if fiber material** option in the **Process Settings Wizard** is selected to run a Fiber orientation analysis.

Setup task	Analysis technology
Core shift simulation on page 21 [3]	
Hot gates	
Overmolding analysis	
Occurrence numbers	
Runner balance constraints [4]	
Cavity/core side mold temperatures [5]	
Clamp force	
Gravity direction	
Valve gates	

Displaying results of first Fill analysis in sequence

If you have more than one Fill result in an analysis sequence, you can choose to display the results for the first one.

When viewing the results of an analysis sequence that contains more than one Fill or Fill+Pack analysis (for example, `Fill+Pack + Cool + Fill+Pack + Warp`), the available Fill+Pack results all relate to the final Fill+Pack analysis in the sequence. To view the results of the initial Fill or Fill+Pack analysis, follow the procedure below.

1 Make sure you have a study with results available.
2 Duplicate the study.
3 Open the duplicate study by double-clicking it.
4 Change the analysis sequence to only include the first analysis in the sequence.
5 Click (**Start Analysis!**)
The results of the initial Fill or Fill+Pack analysis are immediately available.

[3] This is required for a Core-shift analysis.
[4] Only for Runner Balance analyses.
[5] Manual core/cavity mold temperature assignment is only relevant when not performing Cool analysis.

Changing the default viscosity model

1. Click **Home tab > Molding Process Setup panel > Process Settings**.

   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 **Rheological Properties** tab, then select the required shrinkage model in the **Default viscosity model** drop-down list.

   You can click **Edit viscosity model coefficients** to change the viscosity model coefficients.
5. On the same dialog, you can also set whether the extension viscosity model is used.

Fill analysis

Use this dialog to specify settings for a Fill analysis.

Process Settings Wizard dialog—Fill 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 filling phase simulation related process settings for the selected analysis sequence.

NOTE: The Fill analysis process settings normally contains three controls, as per the Fill+Pack analysis sequence process settings: filling control, velocity/pressure switchover and pack/holding control. At the end of the velocity phase the part is only partially filled, according to the filling control and velocity/pressure switchover settings. The remaining portion is filled by the application of packing pressure, which is set using the pack/holding control. This is the reason packing pressure control is required for a Fill analysis.

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.

Mold surface temperature	The temperature of the mold at the plastic-metal interface, where the plastic touches the mold.

Melt temperature	The temperature of the molten plastic, or melt, as it starts to flow into the cavity.
Filling control	Specifies the method by which the filling phase of the analysis is controlled.
Velocity/pressure switch-over	The criteria by which the molding machine will switch from velocity control to pressure control.
Advanced options...	Displays the advanced options for the analysis.
Pack/holding control	Specifies the method by which the pressure phase of the molding process is controlled.
Fiber orientation analysis if fiber material	Enables a Fiber orientation analysis if the material includes fiber.

Process Settings Wizard dialog—Profile 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 filling profile related process settings for the selected thermoset molding analysis sequence.

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

Fast fill analysis

2

The **Fast Fill** analysis provides a quick and simple alternative to the standard Fill analysis.

It can be useful for an approximation of how a part should fill in a situation where accuracy is not a high priority.

NOTE: If you require comprehensive filling results, use the standard Fill analysis.

The Fast Fill analysis and the standard Fill analysis have different characteristics, which are outlined below.

Characteristics

Fast Fill analysis	Fill analysis
Incompressible	Compressible
6 layers across full thickness	Up to 20 layers across full thickness
Aggressive melt front advancement	Conservative melt front advancement
Dynamic convergence criteria	Tight convergence criteria

Compressibility

The incompressible flow in a Fast Fill analysis uses less CPU time than the compressible flow in a standard Fill analysis.

Layers

The Fast Fill analysis uses only 6 layers across the thickness of a flow front. This means that the data concerning heat conduction in the Fast Fill analysis is less detailed than in the standard Fill analysis (which uses up to 20 layers).

Melt front advancement

The aggressive melt front advancement of the Fast Fill analysis results in fewer time steps than the standard Fill analysis, and therefore a faster analysis time.

Convergence criteria

The loose convergence criteria towards the end of the Fast Fill analysis results in less CPU time. In the standard Fill analysis, more nodes are filled near the end of fill, thereby slowing the analysis down.

Fast Fill analysis

Use these dialogs to specify settings for a Fast Fill analysis.

Process Settings Wizard dialog—Fast fill 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 simulation-related process settings for the Fast Fill analysis.

To access this dialog, ensure that you have set the molding process to Thermoplastic Injection Molding, and the analysis sequence to Fast Fill, then click **Analysis > Process Settings Wizard**.

NOTE: The fill-only process normally contains three controls, as per the Fill+Pack process: filling control, velocity/pressure switchover and pack/holding control. At the end of the velocity phase, the part is only partially filled according to the filling control and velocity/pressure switchover settings. The remaining proportion is filled by the application of packing pressure, which is set using the pack/holding control. This is the reason packing pressure control is required for a fill-only analysis.

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

Mold surface temperature	The temperature of the mold at the plastic-metal interface, where the plastic touches the mold.
Melt temperature	The temperature of the molten plastic, or melt, as it starts to flow into the cavity.
Maximum machine clamp force	The maximum allowable clamping force to be used in the Gate Location analysis.
Velocity/pressure switch-over	The criteria by which the molding machine will switch from velocity control to pressure control.
Pack/holding control	Specifies the method by which the pressure phase of the molding process is controlled.

Pack/Holding Control Profile Settings dialog

This dialog is used to enter a packing profile to describe the pressure applied to the cavity during the packing phase of the injection molding process.

The parameters used to define a packing profile differ between different models and makes of injection molding machines. Therefore, a varied selection of input methods are available.

To access this dialog, ensure that you have selected an analysis sequence that includes Fill or Fill+Pack, click (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Fill Settings** or **Fill+Pack Settings** page of the Wizard, then click **Edit profile** next to the **Pack/holding control** option.

You can use shortcuts to edit the input data.

%Filling pressure vs time	Controls the packing phase of the molding cycle as a function of percentage filling pressure versus time.
Packing pressure vs time	Controls the packing phase of the molding cycle as a function of injection pressure versus time.
Hydraulic pressure vs time	Controls the packing phase of the molding cycle as a function of hydraulic pressure versus time.
%Maximum machine pressure vs time	Controls the packing phase of the molding cycle as a function of percentage maximum pressure versus time.

3 Fill+Pack analysis

A Fill+Pack analysis predicts the polymer flow inside the mold during the filling and packing phases. This analysis sequence can be used to determine whether a cavity will be completely filled.

The sequence consists of a Fill analysis followed by a Pack analysis. The Pack analysis uses many results that are calculated in the filling phase to more accurately predict polymer behaviour in the packing phase.

Additional simulation features using the Fill+Pack analysis sequence

Besides analyzing the filling and packing phase of the injection molding process, Fill+Pack analyses also support the following simulation features:

Core shift prediction	If your part is produced with the aid of a core, and you have modeled the core, a Fill+Pack analysis can be used to predict the deflection of the core and the resultant changes in thickness and performance of the part.
2-Shot Sequential Overmolding	If your part is produced in a 2-shot Sequential Overmolding process, a Fill+Pack analysis can simulate the filling and packing of the overmolded component.

Fill+Pack analysis

The analysis sequence consists of a Fill analysis, followed by a Pack analysis. The Pack analysis uses many results calculated in the filling phase to more accurately predict polymer behaviour in the packing phase.

Preparing the model for Midplane Fill+Pack analysis

Before a Midplane mesh analysis can be performed, there are modeling tasks that need to be performed to the model geometry. Compulsory modeling tasks must be performed or the analysis will not run. There are also some optional modeling tasks that can be performed if required.

Compulsory modeling tasks

In order to run a Fill+Pack analysis, the following must be specified for your model:

- A meshed part model.
- Injection nodes specified.

- An oriented mesh.

Optional modeling tasks

Optional boundary conditions that you can assign include:

- Assign top and bottom mold surface temperatures.

Preparing the model for Dual Domain Fill+Pack analysis

Before a surface mesh analysis can be performed, there are modeling tasks that need to be performed to the model geometry. Compulsory modeling tasks must be performed or the analysis will not run. There are also optional modeling tasks that can be performed if required.

Compulsory modeling tasks

- A meshed part model.
- Injection locations specified.
- An oriented mesh (if you want to continue on to a Cool analysis).

Optional modeling tasks

Optional boundary conditions that you can assign include:

- Assign top and bottom mold surface temperatures.

Fill+Pack analysis

Use this dialog to specify settings for a Fill+Pack analysis.

Process Settings Wizard dialog—Fill+Pack 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 filling and packing phase simulation related process settings for the selected 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.

Mold surface temperature	The temperature of the mold at the plastic-metal interface, where the plastic touches the mold.
Melt temperature	The temperature of the molten plastic, or melt, as it starts to flow into the cavity.

Filling control	Specifies the method by which the filling phase of the analysis is controlled.
Velocity/pressure switch-over	The criteria by which the molding machine will switch from velocity control to pressure control.
Pack/holding control	Specifies the method by which the pressure phase of the molding process is controlled
Cooling time	Specify a cooling time, or have it calculated automatically during the Fill+Pack analysis.
Advanced options...	Displays the advanced options for the analysis.
Fiber orientation analysis if fiber material	Enables a Fiber orientation analysis if the material includes fiber.
Birefringence analysis if material data includes optical properties	Produces stress-optical birefringence results if the material has optical properties.
Crystallization analysis (requires material data)	Enables a Crystallization analysis when the material is semi-crystalline and material data includes crystallization morphology parameters.

Fill+Pack Analysis Advanced Options dialog

This dialog is used to specify the Fill+Pack analysis related advanced options for the selected analysis sequence.

To access this dialog, ensure that you have selected an analysis sequence that includes Fill+Pack, click (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Fill+Pack Settings** page of the Wizard, then click **Advanced options**.

Molding material	Select and edit the material to analyze.
Process controller	Allows you to select and edit a process controller to control the injection molding process during the analysis. You can control the filling phase, velocity/pressure switch-over point, pack/holding phase, mold temperature and mold-open time.
Injection molding machine	Select and edit an injection molding machine to simulate your molding machine during the analysis. You can configure the injection unit, hydraulic unit, and clamping unit.

Mold material	Allows you to select and edit the mold material to be used during the analysis. You can specify the density, specific heat, and thermal conductivity of the mold material.
Solver parameters	Allows you to select and edit the solver parameters to be used during the analysis.

Fiber Orientation Solver Parameters dialog

Use this dialog to specify the settings for the solver parameters related to Fiber orientation prediction in a Fill or Fill+Pack analysis sequence.

To access this dialog, ensure that you have selected an analysis sequence that includes Fill+Pack , click (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Fill+Pack Settings** page of the Wizard, select the option **Fiber orientation analysis if fiber material**, then click **Fiber parameters**.

NOTE: All solver parameters have a default value that will be suitable for most analyses.

NOTE: If the shrinkage model for the selected material is set to CRIMS, and the **Use CRIMS** option on the CRIMS Shrinkage Model Coefficients dialog is set to the default (**change solver parameters to be consistent with the CRIMS model**), any changes that you make to the solver parameters above will be overwritten in the analysis. If you want to use non-default settings for these solver parameters, either change the **Use CRIMS** option setting, or select a different shrinkage model.

Calculate fiber orientation using	This option specifies the model used by the Fiber analysis solver to calculate fiber orientation.
Apply fiber inlet condition at	This option specifies whether the fiber orientation calculation begins at the part gate or at the injection location.
Fiber inlet condition	Allows you to specify the inlet boundary condition of the fiber orientation state.
Composite property calculation options	This dialog is used to edit options relating to the prediction of the mechanical properties of the composite, that is, fibers plus polymer matrix.

3D Fill+Pack analysis options

4

Flow is modeled in 3D meshes using Navier-Stokes equations for non-Newtonian viscosity. There are a number of options that you can use to adjust the compromise between speed, accuracy and memory usage.
In particular, turning on or off inertia effects may result in considerably faster analyses.

Simulate inertia effect

The classical, complete Navier-Stokes equations (including inertia) for non-Newtonian viscosity are the slowest but most complete solver option. Inertia is the mass-velocity term in the momentum equations. This means that if fluid has a velocity, it will tend to keep that velocity, unless some other force acts upon it. Other forces can include the viscous forces (stresses) which come from shear deformation. In the particular case of injection molding of polymers, the viscous stresses are very large compared to the inertia terms. This is because of the relatively high viscosity of polymer melt (compared to other fluids such as air) and the narrow cavities through which polymers are injected. This is equivalent to saying: In injection molding, the Reynolds number of the flow is usually much less than one and so inertia terms are not significant.

Generally, use the inertia option when the Reynolds number is expected to be greater than 1. Even then, consider whether your analysis needs to have this accuracy. There might be high velocities in a small gate region, but if the gate is only a small contribution to the total injection pressure, there may be little difference in results with or without inertia.

If the inertia term is removed from the momentum equations, the calculations are simplified and so make some analysis speed improvement. Since in most cases, the dropping of the inertia term will make no difference to injection molding predictions, this option is a good choice for most users. A Navier-Stokes analysis without inertia terms is sometimes called a Stokes analysis. The speed saving is about 10% to 30%.

By default, the inertia option is turned off.

Simulate gravity effect

In most molding situations, the force of gravity is insignificant compared to other forces, such as those brought about by injection and stress. If you need to model the effects of gravity, you can turn on this option.

GPU technology

Using a GPU (Graphics Processing Unit) card enables the analysis to perform numerical intensive calculations on the card itself, resulting in a faster analysis time for 3D flow analyses. When this option is used in conjunction with parallelization, greater speed improvements can be achieved.

By default, GPU technology is turned on.

Support for the legacy Segregated 3D Flow solver

The current Coupled 3D Flow solver has been the default solver since its initial implementation; however, the legacy Segregated 3D Flow solver remained as an option in the solver parameters. In the Autodesk Moldflow 2012 release, the legacy Segregated 3D Flow solver option has been removed.

If studies containing analysis results obtained from the legacy Segregated 3D Flow solver option are imported into the current release, results may still be examined. However, if the analysis is re-run, the current Coupled 3D Flow solver with default solver parameters settings will be used, and a warning message will be displayed to notify you of the change.

3D Fill+Pack analysis types

Before a 3D analysis can be performed, there are modeling tasks that need to be performed to the model geometry.

Preparing the model for 3D Fill+Pack analysis

Compulsory modeling tasks must be performed or the analysis will not run. There are also some optional modeling tasks that can be performed if required.

Compulsory modeling tasks

In order to run a 3D analysis, you must have a 4-noded, tetrahedral-element meshed part model, either

- created in a CAD system and imported,

  or

- created from a surface mesh within Autodesk Moldflow Insight.

Analysis setup options

Select a pressure control point.

Modeling gates

When using 3D analysis technology and Fill+Pack analyses without a modeled gate or runner system, there is a critical significance of implied gate size on the predicted injection pressure.

NOTE: When running a Fill+Pack analysis using 3D analysis technology with beam elements, there must be more than one beam element between the injection node and the tetrahedral elements.

If no gate is modeled (direct injection) the gate size is automatically assigned based on the part geometry, or in the case of small parts the average facet size of the tetrahedral elements on the surface around the injection location. The gate diameter can also be manually specified from the **Solver Parameters** dialog.

Additional simulation features of the Fill+Pack analysis

5

Besides analyzing the packing phase of the injection molding process, the Flow analyses support additional simulation features.

Core shift prediction	If the effect of core is taken into consideration when analyzing a part, a Pack analysis can be used to predict the deflection of the core, and the resultant changes in thickness and performance of the part.
2-Shot Sequential Overmolding	If your part is produced in a 2-shot Sequential Overmolding process, a Fill+Pack analysis can simulate the filling and packing of the first component following by the filling and packing of the overmolded component.

Laminae in a Fill+Pack analysis

6

This topic describes how laminae are split across the part thickness and how laminae are used in an analysis.

The **Number of laminae across thickness** defines the number of layers across the thickness of the part, and can be specified in the Solver advanced options in the Process Settings Wizard. Values that can be specified are 8, 10, 12, 14, 16, 18, and 20.

The following figures show the temperature (X) accross the cavity thickness (Y) at a selected location on the part. The number of results shown on the curve obviously increases with the number of laminae accross thickness.

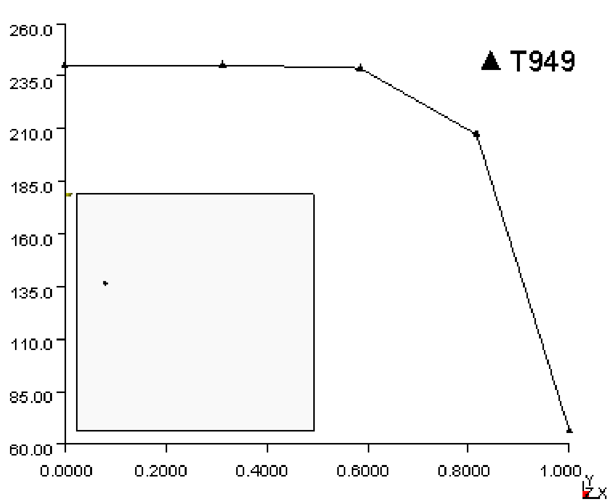


Figure 1: Temperature:XY plot for a part containing 8 laminae accross the thickness of the part.

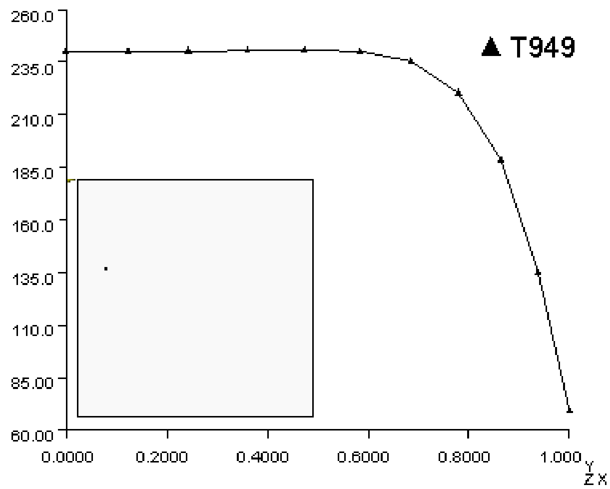
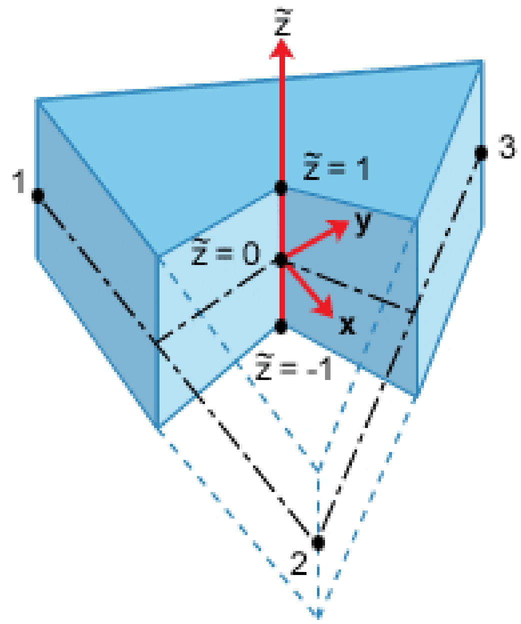


Figure 2: Temperature:XY plot for a part containing 20 laminae accross the thickness of the part.

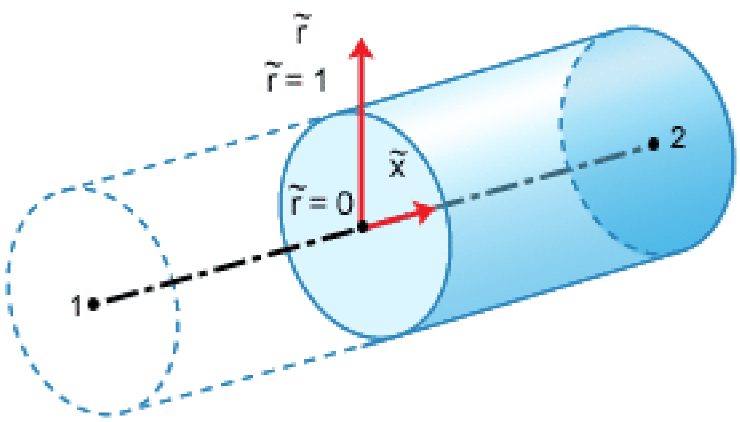
NOTE: For fiber-filled and stress analyses the number of laminae across thickness is set to 20 and cannot be changed.

With triangular elements, 0.0 is at the center plane of the thickness, 1.0 is at the positive mold wall, and -1.0 is at the negative mold wall. The positive side is in the positive direction of the normal to the element, defined by the element connectivity and the right-hand rule.

Note that the local x-coordinate is along the direction of node 1 pointing toward node 2. The y-coordinate lies on the plane defined by nodes 1, 2, and 3, and points toward node 3 perpendicular to the x-coordinate. The z-coordinate (or the element normal) is defined by the cross product of the local x- and y-coordinates.



With axisymmetric one-dimensional elements, only half of the specified layers are used in the analysis. In this case, 0.0 is at the center line of the one-dimensional element, and 1.0 is at the mold wall, as shown below. Note that the local x-coordinate of the one-dimensional element lies along the line defined by nodes 1 and 2, and the direction of the x-coordinate is from node 1 toward node 2.



Depending on the thermal boundary condition across the thickness, the actual number of layers used in the analysis is either the specified value or one-half of the specified number of layers across the thickness. Internally defined, non-uniform layer thicknesses are used in the analysis.

Variables such as temperature, velocity, shear rate, and viscosity are stored at the grid point of each layer. As the number of layers used in the analysis increases, a more accurate numerical solution is expected. However, the CPU time required to complete the analysis increases significantly as the number of layers increase. The disk space required to store the results will also increase as the number of layers increase.

The normalized coordinates of grid points are listed in the table below, where zero is at the center line of the thickness and 1 is at the wall.

Table 1: Number of laminae across the thickness

8	10	12	14	16	18	20
						1.000
					1.000	0.938
				1.000	0.926	0.864
			1.000	0.914	0.840	0.779
		1.000	0.900	0.816	0.743	0.685
	1.000	0.880	0.784	0.706	0.636	0.583
1.000	0.856	0.738	0.653	0.585	0.520	0.474
0.816	0.681	0.577	0.508	0.453	0.397	0.360
0.586	0.477	0.399	0.350	0.310	0.268	0.243
0.313	0.248	0.206	0.180	0.158	0.135	0.123
0.000	0.000	0.000	0.000	0.000	0.000	0.000
0.313	0.248	0.206	0.180	0.158	0.135	0.123
0.586	0.477	0.399	0.350	0.310	0.268	0.243
0.816	0.681	0.577	0.508	0.453	0.397	0.360
1.000	0.856	0.738	0.653	0.585	0.520	0.474
	1.000	0.880	0.784	0.706	0.636	0.583
		1.000	0.900	0.816	0.743	0.685
			1.000	0.914	0.840	0.779
				1.000	0.926	0.864
					1.000	0.938
						1.000

Core shift simulation

7

Core shift is the spatial deviation in the position of the core from its original position in the mold before plastic is injected into the cavity.

It is a frequent problem with long, slender, and not necessarily thin-walled products, such as vials, test tubes, pen barrels. It is also experienced often in molds for thin-walled containers.

Core shift can result in undesirable variations in wall thickness which will affect the final shape and mechanical performance of the part. The Core shift simulation provides detailed information about the movement of the mold core and its interaction with the polymer flow process as the plastic is being injected. Designers can use this information to correct for the core shift phenomenon, for example, by modifying the design of the part, or adjusting process conditions such as the gate location or core/mold temperatures.

There are three main causes of core shift:

- Inaccuracies in the machining or setting of the mold which leads to alignment problems when the mold is closed.
- Deflection of the platens or mold plates due to insufficient strength under the high injection pressures experienced during molding.
- Deflection of the core as a result of pressure differentials on opposing mold walls. These differentials arise as a result of the gate location or variations in part thickness.

The Core shift simulation in Fill+Pack accounts for the third cause described above; the other two causes are difficult to model with a Fill+Pack analysis product such as Autodesk Moldflow Insight.

NOTE: Running a core shift analysis requires an unreserved Performance level license to be available on your license server.

How the Core shift analysis works

The Core shift simulation uses the Stress analysis capabilities of the warpage product to predict the deflection of the core under the loads (pressures) experienced in the molding process. The analysis scheme can be outlined as follows:

- A portion of the cavity is filled (the %volume increment is user-defined in the solver parameters).
- The Fill+Pack analysis provides the pressure load distribution on the core.
- The pressure load data is used to perform a structural analysis of the core.
- The structural analysis determines the core deflection.
- The core deflection is used to calculate the change in part thickness.

- The part mesh is adjusted to account for the change in thickness.
- The above steps are repeated until the cavity is filled.
- If the packing analysis is also being simulated, the structural analysis is also repeated at regular time increments during the packing phase.

For a detailed description of how core shift is simulated, see the following published article:

Bakharev, A., Fan, Z., Costa, F., Han, S., Jin, X. and Kennedy, P., Prediction of Core Shift Effects using Mold Filling Simulation, Soc. Plastics Engineers, ANTEC 04.

Modeling requirements for core shift

The Core shift simulation requires a tetrahedral meshed representation of the core. Constraints need to be applied to the fixed end of the core to prevent rigid body motion in the stress analysis.

Core shift simulation

There are a variety of ways to create a core for a core-shift analysis.

Modeling the core

You can create the core for a Core-shift analysis using an existing model. Alternatively, you can import the core from a CAD program.

If you want the core to extend past the end of the part, use  (**Inserts**) to create a mold insert.

NOTE: Ensure the part uses a Dual Domain mesh. If your part uses a Midplane mesh, convert it to Dual Domain mesh before following the instructions. Make sure the mesh has no errors.

---

1. Create a copy of the study in which you want to create a core.
2. Create a new layer.
3. Select the elements that are touching the core, and assign them to the new layer.
4. Delete the elements that are not part of the core, and the layers that do not contain core elements.
5. Delete all the remaining unused nodes that are not connected to the core elements.
6. Fill any holes in the mesh.
7. Change the properties of all elements on all layers to **Part Surface (Dual Domain)**.
8. Change the mesh type to 3D and remesh the core.

TIP: Use a minimum of 4 elements through the thickness of the mesh.

---

9 Set the **Property Type** of all elements on all layers to **Core 3D**.

TIP: The name of the property assigned will be displayed in the analysis logs. If you have multiple cores, you can apply a separate **Core 3D** property with a different name to the elements in each of the cores.

---

10 You can change the material of the core and the local mold surface temperature control by selecting all elements on all layers, then editing their properties (right-click and select **Properties**, or click **Geometry tab > Properties panel > Edit**.

11 You must set a fixed constraint on the nodes at the fixed end of the core, where it joins to the mold. Select **Boundary Conditions tab > Constraints and Loads panel > Constraints > Fixed Constraint**.

12 Select all the nodes at the fixed end of the core.

TIP: Ensure that the **Select Enclosed entities** option is set (**Geometry tab > Selection panel > Select Enclosed entities**) and that you rotate the part so the nodes you want to select are in a line. This stops unwanted nodes from being selected.

---

13 In the **Input Parameters** section of the **Fixed Constraint** tool, select **Core-shift Analysis** from the **Use constraint in** drop down box, then select **Apply** to apply the fixed constraint.

14 Rename the layers with core elements in them to:

- Core nodes
- Core tetras
- Core constraints

This prevents duplication of existing layer names, and allows you to easily identify the core layers when the core is added to the model of the part.

15 Click then click **Save > Save Study** to save your study.

16 Open the study containing the model of the part, and add the study containing the core model to it.

The model is now ready to be used in a Core-shift analysis.

Modeling the core using mold inserts

If you want the core used in a Core-shift analysis to extend past the end of the part, use the **Create Inserts** tool to create a mold insert.

Alternatively you can create the core using an existing model.

Or, you can import the core from a CAD program.

NOTE: Ensure the part uses a Dual Domain mesh. If your part uses a Midplane mesh, convert it to Dual Domain mesh before following the instructions. Make sure the mesh has no errors.

1 It is easier to use (**Inserts**) when you hide the elements of the part that are not in contact with the core.

   a Create a new layer to hold the non-core elements.

   b Assign all elements not touching the core to the new layer.

2 Click **Geometry tab > Create panel > Inserts** to display the **Create Mold Inserts** tool.

3 Select the elements on the current layer. These should be the only elements that touch the core. Select the direction in which the core is to be projected and specify the length of the core as the projection distance (enter a negative value if projecting in the negative direction of the selected axis), then click **Apply**.

   TIP: If you do not get the expected results, click (**Undo**) and repeat this step.

4 Hide all layers apart from the newly created Mold insert layer created by the **Create Mold Inserts** tool. Rename the Mold insert layer to **Core elements**.

5 Change the properties of the elements on the **Core elements** layer to **Part Surface (Dual Domain)**.

6 Change the mesh type of the elements on the **Core elements** layer to 3D and remesh the core.

   TIP: Use a minimum of 4 elements through the thickness of the mesh.

7 Rename the **New tetras** layer to **Core tetras**. Rename the **New Nodes** layer to **Core Nodes**. This allows you to easily identify the core layers.

8 Set the **Property Type** of the elements on the **Core elements** layer to **Core 3D**.

   TIP: The name of the property assigned will be displayed in the analysis logs. If you have multiple cores, you can apply a separate Core 3D property with a different name to the elements in each of the cores.

9 You can change the material of the elements on the **Core elements** layer and the local mold surface temperature control by selecting the elements on the **Core elements** layer, then editing their properties **Geometry tab > Properties panel > Edit**.

10 You must set a fixed constraint on the nodes at the fixed end of the core, where it joins to the mold. Click **Boundary Conditions tab > Constraints and Loads panel > Constraints > Fixed Constraint**.

11 Select all nodes on on the **Core nodes** layer that are at the fixed end of the core.

TIP: Ensure that the **Select Enclosed entities** option is set (**Geometry tab > Selection panel > Select Enclosed entities**) and that you rotate the part so the nodes you want to select are in a line. This stops unwanted nodes from being selected.

---

12 In the **Input Parameters** section of the **Fixed Constraint** tool, select **Core-shift Analysis** from the **Use constraint in** drop down box, then select **Apply** to apply the fixed constraint.

13 Display all the model layers by selecting them in the **Layers pane**.

14 Right-click the **Mesh Type** icon in the **Study Tasks** pane, then reselect the original mesh type, Midplane or Dual Domain of the part.

The model is now ready to be used in a Core-shift analysis.

Importing a model of the core from a CAD program

You can import the core model from a CAD program.

If you want the core to extend past the end of the part, use (**Inserts**) to create a mold insert.

NOTE: You can prepare the core while it is mesh if the starting point of the core is the mesh of the part, originally exported from Autodesk Moldflow Insight. This ensures the mesh of the core will match the part perfectly. If you are importing the geometry of the core created in a CAD package, it will be difficult to match the surface mesh between the core and the part.

---

1 Import the CAD model of the core into a new study, using a Dual Domain mesh with a similar density to the part model.

2 Double click the mesh icon in the **Study Tasks** pane to create the Dual Domain mesh.

3 Repair the mesh where necessary to ensure it has no errors.

4 Change the properties of the elements on the **Core elements** layer to **Part Surface (Dual Domain)**.

5 Change the mesh type to 3D and remesh the core.

TIP: Use a minimum of 4 elements through the thickness of the mesh.

NOTE: The core mesh and the part mesh must be within the Surface Matching Tolerance value (default: 0.22mm).

---

6 Set the **Property Type** of all elements on all layers to **Core 3D**.

TIP: The name of the property assigned will be displayed in the analysis logs. If you have multiple cores, you can apply a separate **Core 3D** property with a different name to the elements in each of the cores.

---

7 You can change the material of the core and the **local mold surface temperature control** by selecting all elements on all layers, then editing their properties: **Geometry tab > Properties panel > Edit**.

8 You must set a fixed constraint on the nodes at the fixed end of the core, where it joins to the mold. Click **Boundary Conditions tab > Constraints and Loads panel > Constraints > Fixed Constraint**.

9 Select all the nodes at the fixed end of the core.

TIP: Ensure that the **Select Enclosed Entities** option is set (**Geometry tab > Selection panel > Select Enclosed entities**) and that you rotate the part so the nodes you want to select are in a line. This stops unwanted nodes from being selected.

---

10 In the **Input Parameters** section of the **Fixed Constraint** tool, select **Core-shift Analysis** from the **Use constraint in** drop down box, then select **Apply** to apply the fixed constraint.

11 Rename the layers with core elements in them to:

- Core nodes
- Core tetras
- Core constraints

This prevents duplication of existing layer names, and allows you to easily identify the core layers when the core is added to the model of the part.

12 Click then click **Save > Save Study** to save your study.

13 Open the study containing the model of the part, and add the study containing the core model to it.

The model is now ready to be used in a Core-shift analysis.

Representative shear rate derivation

8

The bulk shear rate, or representative shear rate is derived from the wall shear stress and the fluidity, which are quantities calculated during an analysis.

A representative viscosity, η_{rep} is calculated from the fluidity, S , and the thickness of the part, h , as: $S \equiv \int_0^h \frac{z^2}{\eta} dz = \frac{h^3}{3\eta_{rep}}$

The representative shear rate is then calculated from the wall shear stress and the representative viscosity: $\dot{\gamma} = \frac{\text{wall shear stress}}{\eta_{rep}}$