Autodesk® Moldflow® Insight 2012

AMI Flow Analysis Results

Autodesk®

Revision 1, 23 March 2012.

This document contains Autodesk and third-party software license agreements/notices and/or additional terms and conditions for licensed third-party software components included within the product. These notices and/or additional terms and conditions are made a part of and incorporated by reference into the Autodesk Software License Agreement and/or the About included as part of the Help function within the software.

Contents

Flow results

1

This help topic specifies the results generated for analyses that include Fill, Fill+Pack, or Standalone Pack analysis sequences.

These results are generated for material with or without fiber or filler content.

Before running an analysis, you should pay close attention to the edge length of the mesh around high curvature areas on your model, and make sure they are not too coarse. It is recommended you mesh with a smaller edge length allowing the mesh to approximate the corners correctly.

Text based results

The following table lists the text results that are created for each these analyses.

Results
Analysis Log
Results Summary
Analysis Check
Machine setup result on page 41

Graphical results

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

- Midplane
- Dual Domain
- 3D

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

NOTE: Overmolding results are available only if you are using an overmolding process. Thermoplastics overmolding analyses produce two sets of Fill analysis results, one for the first component and another for the overmolded component.

Result	Available for analysis type
%Shot weight result on page 6	Midplane, Dual Domain (icons)

Result	Available for analysis type
Air traps result on page 7	
Air traps, including air vents result on page 9[1]	
Average velocity result on page 10	
Bulk temperature result on page 12	
Bulk temperature, elemental result on page 13	
Bulk temperature, nodal result on page 14	
Bulk temperature at end of fill result on page 15	
Clamp force result on page 17	
Clamp force centroid result on page 18	
Density result on page 19	
Displacements, core result on page 20	
Displacements, final shift result on page 21	
Extension rate result (3D) on page 22	
Fill time result on page 24	
Flow rate, beams result on page 26[2]	
Time to reach ejection temperature result on page 82	
Frozen layer fraction result on page 29	
Frozen layer fraction at end of fill result on page 31	
Frozen pressure result on page 32	
Hold pressure result on page 35	
Grow from result on page 33	
In-cavity residual stress in first principal direction result on page 37	

[1] This result is available only when the option to Perform venting analysis is selected in the solver parameters.

[2] The Flow rate, beams result will appear in any Fill analysis where 1-dimensional elements are modeled.

Result	Available for analysis type
In-cavity residual stress in second principal direction result on page 38	
Interface heat flux (3D Overmolding) result on page 39	
Orientation at core result on page 42	
Orientation at skin result on page 43	
Orientation at bottom skin result on page 45	
Orientation at top skin result on page 47	
Polymer fill region result (3D) on page 50	
Pressure result on page 51	
Pressure at end of fill result on page 53	
Pressure at injection location result on page 56	
Pressure at velocity/pressure switchover result on page 57	
Re-melt zone, overmolded components (3D Overmolding) result on page 58	
Re-melt zone, part insert result on page 59	
Real thickness, cavity result on page 16	
Ram speed, recommended result on page 60	
Shear rate result on page 61	
Shear rate result (3D) on page 63	
Shear rate, bulk result on page 64	
Shear rate, maximum result on page 65	
Shear stress at wall result on page 66	
Sink marks, index result on page 68	
Sink marks, depth result on page 70	
Sink marks estimate result on page 71	
Temperature result on page 73	

Result	Available for analysis type
Temperature result (3D) on page 74	
Temperature at flow front result on page 76	
Throughput result on page 81	
Time to reach ejection temperature result on page 82	
Unfilled cavity result on page 83	
Velocity result on page 84	
Velocity result (3D) on page 85	
Viscosity result on page 87	
Vent region pressure result on page 86[1]	
Volumetric shrinkage	
Average volumetric shrinkage result on page 88	
Volumetric shrinkage at ejection result on page 92	
Weld lines result on page 94	
Weld and meld lines result on page 97	

Fill analysis results

You can generate results for a Fill analysis on a thermoplastic material.

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.

%Shot weight result

2

The %Shot weight result shows the total shot weight, as a percentage of the total part weight, at various time-steps during the filling analysis.

Using this result

Because the total part weight changes with time, the %Shot weight result measures the total part weight, as a percentage of the total part weight, at various time-steps during the filling analysis.

The total part weight is determined from the room-temperature density and the total volume defined in the finite-element mesh. From this information, you can decide if removing the holding pressure will influence the shot weight. The percentage runner weight is also displayed with reference to the total part weight. The economics of the runner design can be assessed by considering its percentage weight in the total shot.

Things to look for

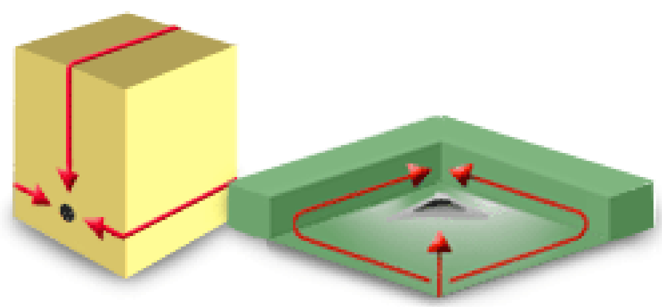
- Pressure result.
- Pressure at end of fill result.

3 Air traps result

An air trap occurs where the melt traps and compresses a bubble of air or gas between two or more converging flow fronts, or between the flow front and the cavity wall. Typically, the result is a small hole or a blemish on the surface of the part. In extreme cases, the compression increases the temperature to a level that causes the plastic to degrade or burn.

Air traps are often due to converging flow fronts caused by racetrack or hesitation effects, or by non-uniform or non-linear fill patterns. Even when the part has balanced flow paths, inadequate venting can cause air traps to occur at the ends of flow paths.

For a Midplane or Dual Domain model, the Air traps result shows a thin, continuous line wherever an air trap is likely to occur. For 3D models, it displays a contour plot indicating the probability of an air trap occurring.



Using this result

The Air traps result shows how severe air traps will be and where they are likely to occur on the part. Air traps may be acceptable if they occur on a surface that does not have to be visually perfect .

The Fill time result is used in conjunction with the Air traps result to confirm the filling behavior and assess the likelihood of air traps appearing.

The air trap plot is used to determine the probability of an air trap occurring at a particular location. A higher value indicates a higher probability that an air trap will occur, and a lower value indicates a lower probability that an air trap will occur.

The Air traps result can reveal the following problems in your part:

- Burn marks caused by air in an air trap, which ignites under pressure and burns the plastic.
- Short shots caused by the incomplete filling of the part; if an air trap is not vented and not compressed quickly enough to cause a burn mark, it may cause a short shot or leave bubbles of air or gas in the plastic part.

- Other surface blemishes caused by air traps.

Things to look for

The following methods can be used to prevent air traps:

- Use flow leaders or deflectors.
- Increase the injection speed to eliminate air traps caused by converging flow fronts and hesitation.
- Decrease the injection speed to reduce air traps caused by poor venting, and to prevent burn marks.
- Decrease the part wall thickness ratio to reduce racetracking.
- Move the injection locations so that the air traps form in areas that are easy to vent, such as the parting plane.

Air traps, including air vents result 4

The Air traps, including air vents result shows the locations where air traps are likely to occur, including specified venting analysis (air vent) locations.

This result is generated by a Fill or Fill+Pack analysis sequence for Thermoplastics Injection Molding, Thermoplastics Overmolding, Reactive Molding or Microchip Encapsulation molding processes using 3D analysis technology, when the option to **Perform venting analysis** is enabled.

You can control how this result is displayed by editing the plot properties.

Using this result

Use the Air traps, including air vents result to aid in selecting appropriate locations for air vents. If the specified venting analysis (air vent) locations coincide with predicted air trap locations, the specified locations are suitable to place air vents in the mold.

TIP: Use the Air traps result to identify preliminary areas for setting venting analysis locations. Then enable the option to perform venting analysis and run the analysis again.

Things to look for

To optimize the placement of air vents, try one of these options.

- Move the injection location(s) so that the air traps form in easy-to-vent areas.
- If no air trap is predicted near a specified venting analysis location, an air vent may not be needed at that location to remove trapped air from the mold.
- If an air trap is predicted near but does not coincide with a specified venting analysis location, move the venting analysis location to more closely match the location of the trapped air to be vented.

Average velocity result

5

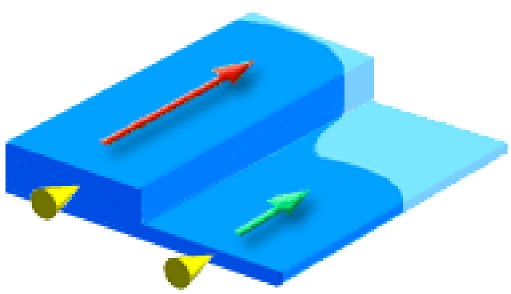
The Average velocity result shows the average magnitude of velocity of the polymer inside the mold cavity over time.

The magnitude of the flow velocity is a straight average through the thickness (but only the melt is considered, not the frozen layer).

NOTE: The Average velocity result is an intermediate result, meaning its animation by default is through time and the scale by default is the minimum to maximum of the entire range of the result.

Using this result

The Average velocity result can be used to determine areas with a high flow rate. High velocity values for a particular model section could indicate a high flow rate, meaning there could be filling problems such as overpacking or flash. This could also mean that polymer flow is unbalanced, where polymer flows fast through one section and flows slowly through another section of the part.



NOTE: Right-clicking the result in the **Study Tasks** pane and selecting **Properties** allows you to plot velocity in either the x, y, or z directions, and to specify the velocity dart length.

When the Average velocity result is created as an XY plot, the direction in which velocity is displayed is indicated by the legend values in the top right. For example, E3_X means x component of element 3.

Combined with fill time information, velocity result plots can help you to determine gate locations, runner sizes, and part thickness to achieve a balanced mold and runner design.

Things to look for

- Overpacking.
- Hesitation.
- Racetrack Effect.
- Unbalanced Flow.

Bulk temperature result

6

Bulk temperature is used to indicate the weighted average temperature across the thickness.

The temperature of polymer melt changes not only with time and location, but also with thickness during the entire injection molding cycle. It is difficult to illustrate all these changes in a single display. For these reasons, Bulk temperature is used. Bulk temperature has more physical significance than a simple average temperature as the polymer melt flows. It also represents the energy that is transported through a particular location.

NOTE: The Bulk temperature result is an intermediate result, meaning its animation by default is through time and the scale by default is the minimum to maximum of the entire range of the result.

Using this result

Bulk temperature is a velocity-weighted average temperature when the polymer is flowing and a simple average temperature when the flow stops. For each element, a plot of bulk temp versus time shows that the switch-over from bulk temperature to average temperature gives a smooth curve. Uniform bulk temperature distribution during filling is desirable for mold design.

Bulk temperature displays are an alternate way to examine the flow distribution. Areas with continuous flow (heat convection) typically have a higher bulk temperature. The bulk temperature drops quickly when the flow stops in that area.

Hot spots are indicated on the bulk-temperature contour or data-shaded plots during filling. Hot spots are due to excessive viscous heating during the filling stage.

If the maximum bulk temperature is close to the degradation temperature, consider redesigning the part geometry near the hot spot or changing the process conditions.

Differential temperature can also cause non-uniform shrinkage and warpage.

Things to look for

- Hot spots—try to acheive more uniform cooling

7 Bulk temperature, elemental result

The Bulk temperature, elemental result is used to indicate the weighted average temperature across the part thickness.

It is a velocity-weighted average temperature when the polymer is flowing and a simple average temperature when the flow stops.

Using this result

Bulk temperature displays are an alternate way to examine the flow distribution. Areas with continuous flow (heat convection) typically have a higher bulk temperature. The bulk temperature drops quickly when the flow stops in that area.

Hot spots are indicated on the bulk-temperature contour or data-shaded plots during filling. Hot spots are due to excessive viscous heating during the filling stage.

If the maximum bulk temperature is close to the degradation temperature, consider redesigning the part geometry near the hot spot or changing the process conditions.

Things to look for

- Uniform bulk temperature distribution during filling is desirable for mold design.

Bulk temperature, nodal result 8

The Bulk temperature, nodal result is created when running a Reactive Molding Fill analysis or a Microchip Encapsulation Fill analysis.

It is used to indicate the weighted average temperature across the part thickness. It is a velocity-weighted average temperature when the polymer is flowing and a simple average temperature when the flow stops.

Using this result

Bulk temperature displays are an alternate way to examine the flow distribution. Areas with continuous flow (heat convection) typically have a higher bulk temperature. The bulk temperature drops quickly when the flow stops in that area.

Hot spots are indicated on the bulk-temperature contour or data-shaded plots during filling. Hot spots are due to excessive viscous heating during the filling stage.

If the maximum bulk temperature is close to the degradation temperature, then consider redesigning the part geometry near the hot spot or changing the process conditions.

Things to look for

- Uniform bulk temperature distribution during filling is desirable for mold design.

Bulk temperature at end of fill result

9

Bulk temperature is used to indicate the weighted average temperature across the thickness.

The temperature of polymer melt changes not only with time and location, but also with thickness during the entire injection molding cycle. It is difficult to illustrate all these changes in a single display. For these reason, Bulk temperature is used. Bulk temperature represents the energy that is transported through a particular location. It has more physical significance than a simple average temperature as the polymer melt flows.

Using this result

Bulk temperature is a velocity-weighted average temperature when the polymer is flowing and a simple average temperature when the flow stops. For each element, a plot of bulk temp versus time shows that the switch-over from bulk temperature to average temperature gives a smooth curve. Uniform bulk temperature distribution during filling is desirable for mold design.

By looking at the bulk temperature distribution at the end of fill, the effect of the next phase, packing pressure, can be estimated. In areas where the bulk temperature is close to transition temperature, the effect of the packing pressure can be low. In areas where bulk temperature is close to melt temperature, packing can be effective. The cooling time, however, can be increased.

Bulk temperature displays are an alternate way to examine the flow distribution. Areas with continuous flow (heat convection) typically have a higher bulk temperature. The bulk temperature drops quickly when the flow stops in that area.

Hot spots are indicated on the bulk-temperature contour or data-shaded plots during filling. Hot spots are due to excessive viscous heating during the filling stage.

If the maximum bulk temperature is close to the degradation temperature, consider redesigning the part geometry near the hot spot or changing the process conditions.

Differential temperature can also cause non-uniform shrinkage and warpage.

The bulk temperature at end of fill will have an effect on the next phase of the analysis.

Things to look for

- Hot spots—try to acheive more uniform cooling

Real thickness, cavity result 10

The Real thickness, cavity result is created only if you have modeled a core, and selected the **Perform core shift analysis** option in the Fill analysis advanced options.

It shows the part (cavity) thickness during the filling and packing phases of the molding cycle, taking into account the displacement of the core. The default display shows the thickness values in the part, at the time indicated on the top right corner of the display.

Using this result

As the core deflects, the area of the part towards which the core deflects will be thinner, and the area of the part away from the deflected core will be thicker. The variations in cavity thickness can be seen by animating the result using the **Animation** toolbar.

By default, this result shows the thicknesses throughout the part by means of the **Animation** plot type.

You can also investigate the change in thickness over time at various locations on the part by creating a new XY plot or path plot of this result.

Things to look for

- Excessive changes in thickness while the part is filling, which can lead to problems such as unbalanced flow.
- Excessive changes in thickness at the end of packing, which can lead to part warpage and a reduction in structural performance.

Clamp force result 11

The Clamp force result is a time-series result which shows the force of the mold-clamp over time.

The clamp force is the resultant value of the pressure distribution over the entire part. It is a history of the force resultant from filling and packing pressure that acts to open the mold.

Using this result

The clamp force is a function of injection pressure and the projected area of the part. The projected area is the area of the model projected onto the XY plane. A good clamp force history result should show that the maximum clamp force applied is less than approximately 80% of the machine limit, allowing the remaining 20% as a safety factor.

There are other factors that affect the necessary safety margin such as sliding cores, guide pins and other tool dependent pre-loads required by the tool configuration. If your design requires these, then you should allow a larger safety factor.

NOTE: For the clamp force calculations to be correct, the model orientation must be such that the clamp force is applied along the Z axis direction. The clamp history result can produce misleading results if your model has overlapping surfaces on the XY plane, as the clamp force for these surfaces is added.

Things to look for

- Pressure spikes from areas in the part that are difficult to fill.
- Clamp force exceeding the guideline.

TIP: Find the maximum value by clicking **Advanced** in the **Process Settings Wizard**, selecting an injection molding machine, and viewing the **Clamping Unit** tab.

Clamp force centroid result 12

The Clamp force centroid result shows the center of the clamp force applied on the part, or the center of mass or gravity.

NOTE: You must ensure the model is correctly oriented to generate the correct result. The clamp force direction is + Z direction in the coordinate axis.

Using this result

The Clamp force centroid result displays the center of the clamp force pressure in your mold design, indicated by the black arrow on the part. The direction of the arrow indicates the mold opening direction.

NOTE: The centroid is recorded at the time of the highest clamp force.

Things to look for

- The clamp force centroid should be located in the center of the part to indicate an even clamping force.
- The arrow should indicate the mold opening direction.

Density result

13

The Density result is generated from a Fill analysis using 3D analysis technology, and shows the density of the tetrahedral elements before shrinkage occurs, at the time the result file was written.

The program measures the density of the material at this time and uses this as the starting point for calculation of the volumetric shrinkage and warpage of the part.

Using this result

The Density result determines whether the part was uniformly packed with polymer during the packing phase. Variations in polymer density can indicate problems with the part.

Once the material has filled the mold cavity and the packing phase has begun, material flow is driven by the variation of density across the part. If one region of a part is less densely packed than an adjacent region, then polymer will flow into the less dense region until equilibrium is reached. This flow will be affected by the compressibility and thermal expansion of the melt in a similar way to which the flow is affected by these factors in the filling phase.

Accurate simulation of the material flow during the packing phase uses the material's pvT characteristics (density variations with pressure and temperature, compressibility, and thermal expansion data) and material viscosity data.

NOTE: Once the pressure at a node reaches zero or the temperature reaches the transition temperature (whichever comes first), the density is recorded. This is one of the initial conditions for Warp analysis. Using this density result to predict the weight of the final part is misleading as the density of the part will change as the part further cools.

Things to look for

- Variations in polymer density
- Hesitation during the filling process
- Overpacking

Displacements, core result 14

The Displacements, core result is created only if you have modeled a core, and the **Perform core shift analysis** option in the Fill analysis advanced options has been turned on.

It shows the deflections of the core during the filling and packing phases of the molding cycle. The default display shows the final deflections of the core at the time indicated in the top-right corner of the display. The magnitude of the displacements is exaggerated by the scale factor set in the plot properties (typically 10).

TIP: This result is best viewed by displaying the layer containing the core tetrahedral elements, and hiding the layer(s) containing the part model.

Using this result

The magnitude and distribution of pressures on the core vary considerably during filling and packing, therefore, the displacement of the core will also vary with time. These effects can be seen by animating the result using the **Animation** toolbar. The maximum value on the legend is the maximum displacement that occurred during the filling and/or packing phases.

By default, the result shows the net displacements throughout the core by means of the **Animation** plot type.

You can also:

- Investigate the displacement components in a specific global or local axis direction using the options on the **Deflection** tab of the **Plot Properties** dialog.
- Investigate the change in displacement over time at various locations on the core by creating a new XY plot, path plot or probe plot of this result.

Things to look for

- Excessive core displacement during the filling phase, which can lead to problems such as unbalanced flow. Typically, the displacement of the core will be at its greatest at the end of the filling phase, and there will be some relaxation back to the original position during the packing phase.
- Excessive core displacement at the end of packing, leading to frozen-in variations in thickness in the part. These variations in thickness can lead to warpage of the part and a reduction in structural performance.

Displacements, final shift result

15

The Displacements, final shift result is created only if you have modeled a core, and the **Perform core shift analysis** option in the Fill analysis advanced options has been turned on.

It shows the final deflection of the core at the end of the filling and packing phases of the molding cycle. The magnitude of the displacements is exaggerated by the scale factor set in the plot properties (typically 10).

TIP: This result is best viewed by displaying the layer containing the core tetrahedral elements, and hiding the layer(s) containing the part model.

Using this result

By default, the result shows the net displacements throughout the core.

You can also investigate the displacement components in a specific global or local axis direction using the options on the **Deflection** tab of the **Plot Properties** dialog.

Things to look for

- Excessive final core displacement, leading to frozen-in variations in thickness in the part. These variations in thickness can lead to warpage of the part and a reduction in structural performance.

Extension rate result (3D) 16

The Extension rate (3D) result is generated from Fill and Microchip Encapsulation analyses using 3D analysis technology, and shows the rate of extension in the mold-cavity at the time the result was written.

Using this result

The extension rate represents the amount of elongation the polymer undergoes as it passes through a change in thickness. Typically this is strongest at gate regions where the flow rate is high and the thickness changes can be large. A contraction is a reduction in thickness along the flow direction, and results in a positive extension (elongation) of the melt. Such extension occurs at the entrance to a gate. Inversely, a negative extension (compression in the flow direction) is experienced by the melt as it passes through an expansion (increase in thickness) such expansion occurs where a narrow gate connects to a thick part of the cavity. Fiber filled materials undergo significant orientation in regions of strong extension flow.

The Extension rate result can be used to see where the flow is undergoing a significant elongation deformation. This will have implications for understanding the additional pressure drop which occurs at gates (sometimes called the entrance pressure loss). The extension rate result is only available on tetrahedral elements. It is not available on beam elements.

When the Extension rate (3D) result is displayed, scale the result using **Results tab > Properties panel > Plot Properties** and the **Scaling** tab.

The extension rate is equal to zero in a part with uniform thicknesses. When material flows through a contraction in the part, extension rate is positive, and when material flows through an expansion, extension rate is negative.

TIP: Use cutting planes to view the extension rate.

Things to look for

- An extension rate of 200 or less is acceptable. If the extension rate is higher than 200, the effects of extensional viscosity need to be considered, as the extensional viscosity may significantly increase the injection pressure required to fill the part. If the material you have selected has juncture loss coefficients data, the effect of extensional viscosity has been incorporated in the result.
- Local thickening at the end of flow or in thin sections can be used to reduce extension rate.

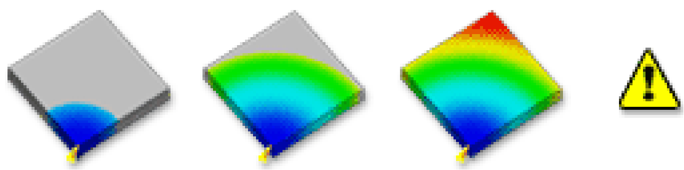
- Extension rate is related to flow kinematics and geometry. In injection molding sense, extension rate is more dependent on flow rate and geometry. The material property may influence the extension rate, but not dependent upon it. It is more like that the viscosity is influenced by extension rate.

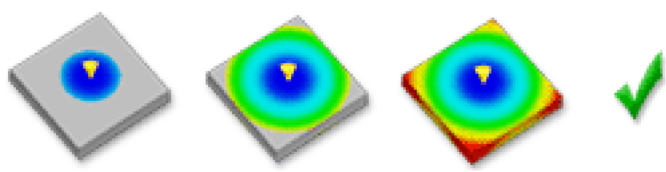
Fill time result

17

The Fill time result shows the position of the flow front at regular intervals as the cavity fills.

The following diagrams show the contour colors that represent the flow of plastic into the part. All regions with the same color are filled simultaneously. The result is dark blue at the start of the injection, and the last areas to fill are red. If the part is a short shot, the section that did not fill has no color.





Using this result

The flow pattern is balanced in a part with a good fill time result, which means the following:

- All flow paths finish at the same time and reach the edges of the model simultaneously. In the previous diagram, each flow path should end with red contours.
- The contours are evenly spaced and indicate the speed at which the polymer is flowing. Widely-spaced contours indicate rapid flow; narrow contours indicate that the part is filling slowly.

Things to look for

Short shot On the Fill time result, a short shot will appear as translucent. Check the ends of the flow paths for any translucent areas. For models analyzed using 3D analysis technology, you can also use the Unfilled cavity result to see if there are any unfilled sections in the interior of the part.

Hesitation	If a Fill time result shows a section where the contours are very closely spaced, hesitation may have occurred. Hesitation can cause a short shot if a thin section freezes off before the part is completely filled.
Overpacking	If a Fill time result shows that one flow path finishes before others do, it may indicate overpacking. Overpacking can cause high part weight, warpage, and non-uniform density distribution throughout the part.
Weld lines	Overlay the Weld line result on the Fill time result to confirm their presence. Weld lines can cause structural and visual defects.
Air traps	Overlay the Air trap result on the Fill time result to confirm their presence. Air traps can cause structural and visual defects.
Racetrack effect	The racetrack effect can cause air traps and weld lines. Check the location and number of air traps and weld lines.

18 Flow rate, beams result

The Flow rate, beams result shows the amount and rate of polymer that is delivered through runners into the mold cavity, allowing you to optimize the runner system design.

NOTE: The Flow rate, beams result is an intermediate result, meaning its animation-by-default is through time and the scale-by-default is the minimum to maximum of the entire range of the result.

Using this result

The Flow rate, beams result is calculated from the product of the average velocity and the cross-sectional area of the runner. This information is very useful for designing the runner system, especially when using a cavity with more than one gate. The inlet flow from the nozzle is distributed among all runner branches during filling. The flow distribution is dynamically adjusted, depending on the resistance from each branch.

For example, if the melt front of one branch reaches a thin section, the flow resistance increases so that the flow rate in that branch is dynamically reduced. At the same time, the flow rate in other branches must be increased to maintain the total mass balance.

From the plots of flow rate versus time at each runner branch, the amount of polymer that is actually delivered from that branch can be calculated. Runners that deliver an insignificant amount of polymer should be either re-sized or eliminated.

Things to look for

- Hesitation
- Unbalanced flow

Flow rate result, shown in the analysis log

19

You will notice that the flow rate shown in the analysis log does not typically reach the user-specified flow rate. Material compressibility accounts for the difference in flow rate input and output.

Material compressibility affects flow rate

For Fill analyses, the effects of material compressibility in the barrel are calculated.

The compressibility of the material in the barrel can be significant. The polymer density in the barrel increases as the ram moves forward and the barrel pressure increases. The flow rate based on the ram movement is larger than the flow rate out of the nozzle, and the polymer in the barrel has to be taken into consideration if the predictions are to be used for machine control.

During an analysis, the barrel volume is added to the model in the calculation, with the melt in the barrel assumed to be isothermal at melt temperature. As the ram advances, the melt is transferred from the barrel to the sprue/runner/cavity, and the melt in the barrel is reduced accordingly.

User input and the compressibility calculation

The compressibility calculation does not depend on user-specified barrel dimensions if the following settings were used:

Fill analysis settings > Filling control > Ram Speed Profile > % shot volume vs % flow rate, or

Fill analysis settings > Filling control > Ram Speed Profile > % stroke vs % ram speed

The compressibility calculation does take into account the user-specified shot size and machine screw diameter if any other ram speed profile setting was used.

If the the shot size and machine screw diameter were not specified, then an automatic stroke volume of 120% is used. This value is considered sufficient to fill the part and takes the barrel effect into account.

NOTE: The flow rate represents the polymer as it compresses. Zero compressibility (100% flow rate) does not closely simulate the behaviour of the polymer in the molding machine.

To achieve a 100% flow rate with virtually zero compressibility, set the following pvT Properties in the Thermoplastics material dialog.

Property	Value
Melt density	1
Solid density	1
b6	0
b2m	1.01e-009
b3m	4.99e+010
b4m	1.01e-007
b2s	1.01e-009
b3s	4.99e+010
b4s	1.01e-007
b7	0
b8	0
b9	0

Frozen layer fraction result 20

The Frozen layer fraction result shows the thickness of the frozen layer as a fraction of the part thickness.

The values of this result range from zero to one. A higher value represents a thicker frozen layer, a higher flow resistance, and a thinner polymer melt or flow layer. A polymer is considered to be frozen when the temperature falls below the transition temperature (Ttrans).

NOTE: The Frozen layer fraction result is an intermediate result, which is animated through time by default. The default scale for this result is from the minimum to the maximum of the entire range of the result.

During filling, the frozen layer should maintain a constant thickness in areas with continuous flow because the heat loss to the mold wall is balanced by the hot melt coming from upstream. When the flow stops, the heat loss through the thickness dominates, resulting in a rapid increase in the thickness of the frozen layer.

Frozen-layer thickness has very significant effects on the flow resistance. The viscosity exponentially increases with decreasing temperature. The thickness of the flow layer is also reduced as the thickness of the frozen layer increases.

The effect of the thickness reduction can be roughly estimated with the definition of fluidity, as with representative shear rate. The fluidity is proportional to the cubic power of the part thickness. A 50 percent reduction in part thickness reduces the fluidity by a factor of eight, or increases the flow resistance by a factor of eight. A 50 percent reduction in thickness in runners reduces the fluidity by a factor of 16.

Using this result

Ideally the part freezes uniformly and as quickly as possible. The Frozen layer fraction result is used in conjunction with the Time to reach ejection temperature result to locate problem areas in the mold. The Frozen layer fraction result can reveal the following problems in your part:

- Excessive high pressure is required to fill parts in which hesitation occurs early in the filling stage. The flow layer becomes very thin in areas of hesitation, which are filled last.
- If the Time to reach ejection temperature values for the part as a whole appear high, then general measures for reducing the excessive cycle time, such as adjusting the mold and melt temperatures, may be required.
- Areas of the part taking longer to cool indicate hot spots.

Things to look for

If hot spots occur, look for ways in which the cooling circuit design can be improved.

Frozen layer fraction at end of fill result 21

The Frozen layer fraction at end of fill result represents the thickness fraction of the frozen layer at the end of filling.

It ranges from 0.0 to 1.0. A higher value indicates a thicker frozen layer (or thinner flow layer) and higher flow resistance. A polymer is considered frozen when the temperature falls below the transition temperature (Ttrans).

During filling, the frozen layer should maintain a constant thickness for those areas with continuous flow, because the heat loss to the mold wall is balanced by the hot melt coming from upstream. Once the flow stops, the heat loss through the thickness is completely dominant in that area. A rapid increase in the thickness of the frozen layer results.

Frozen-layer fraction has very significant effects on the flow resistance. The viscosity exponentially increases with decreasing temperature. The thickness of the flow layer is also reduced as the thickness of the frozen layer increases.

The effect of the thickness reduction can be roughly estimated with the definition of fluidity, as with representative shear rate. The fluidity is proportional to the cubic power of the part thickness. A 50% reduction in part thickness reduces the fluidity by a factor of eight (or increases the flow resistance by a factor of eight). Moreover, a 50% reduction in thickness in runners reduces the fluidity by a factor of 16. It is not surprising that excessive high pressure is required to fill parts in which hesitation occurs early in the filling stage. The flow layer becomes very thin in areas of hesitation, which are filled last.

Using this result

The frozen layer fraction generally will be very low near the injection location and the end of fill. The maximum frozen layer fraction at the end of fill should be less than 0.20–0.25. Higher values will make the part difficult to pack out. Areas of the mold that are filled early in the cycle, but have little subsequent flow, normally have the highest frozen layer fraction.

Things to look for

- None of the part should have a frozen layer fraction higher than 0.20–0.25 at the end of fill. A faster injection time will reduce the frozen layer fraction.
- Time to reach ejection temperature result.
- If hot spots are occurring, look for ways to change the cooling circuit design.

Frozen pressure result

22

The Frozen pressure result shows the maximum pressure reached inside the mold from the time an element freezes until the time the result was written.

This result is generated from a Pack analysis using Midplane or Dual Domain analysis technology, and Overmolding analysis.

NOTE: Elements that have not frozen at the end of the simulation contain no data.

Using this result

The Frozen pressure result should show zero pressure at and after the time to reach ejection temperature, in all elements. Differences in pressure at the time to reach ejection temperature indicate stresses in the part.

Things to look for

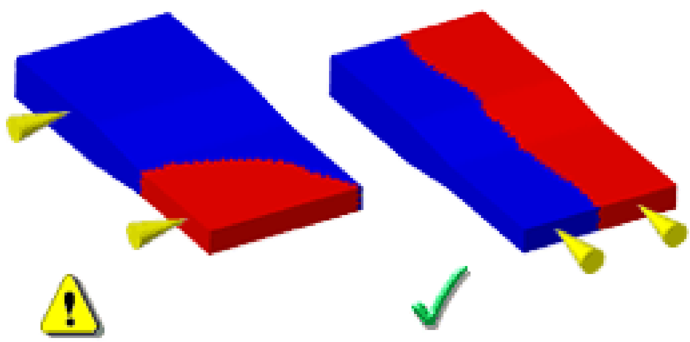
- Pressure should be zero at the extremities of each flow path at the end of filling.
- Pressure variation will produce shrinkage variation and lead to high stresses and/or distortion in the part.

23 Grow from result

The Grow from result, which is produced by a Fill analysis, shows the areas that are filled from each gate in multi-gated parts so that you can determine whether the flow within the part is balanced.

NOTE: You need to assign the “Gate” property to gate element(s) in order to have the correct Grow from result generated.

A Grow from result that shows uniform filling will be balanced. A Grow from result that shows that different gates are trying to fill the same section of the part, will be unbalanced. Non-uniform filling (left) and uniform filling (right) are shown in the following diagrams.



The Grow from result uses numbers to identify the gate in which the polymer was injected. When the Fill analysis is complete, the Grow from result indicates the filling pattern of the part and shows which areas of the part were filled from which gate.

Using this result

A Gate Location analysis can determine more appropriate gate locations on your model, and the Grow from result can be improved in a number of ways. Alterations usually have side effects so after making changes, you should check to see whether other problems have been caused.

Problem	Resolution
Unbalanced flow patterns can be caused by: ■ polymer filling more quickly from one gate	■ Change the injection locations to achieve a balanced fill pattern

Problem	Resolution
■ polymer from both gates trying to fill the same section in the part	■ Redesign the geometry of the model in the problematic area.
A weld or meld line is in a sensitive area.	■ Alter gate positions. ■ Change part thickness. ■ Increase melt and mold temperature. ■ Increase ram speed. ■ Optimize runner system design.

Things to look for

A result that shows an equal distribution of colors will also have a uniform filling pattern and be balanced. If the distribution of colors is not equal, then the flow is unbalanced.

24 Hold pressure result

The Hold pressure result shows the maximum pressure in each area, calculated during the packing phase (from the end of fill).

A big variation in the maximum hold pressure indicates that pressure is not transferred to the extremities of the cavity during the packing phase.

This result is generated from Pack analyses using Midplane or Dual Domain analysis technologies, and Overmolding analyses.

Differences in hold pressure may be caused by the following:

- a poorly designed part with thin sections that freeze early
- an inappropriate gate size, shape or position
- not enough packing pressure
- not packing the cavity for a long enough period

When areas freeze with different pressures, differential shrinkage effects occur, contributing to increased warpage.

Using this result

Use the Hold pressure result to ensure that packing pressure applied at the injection locations is being transmitted throughout the cavity. A uniform pressure distribution will result in even packing as the part freezes, resulting in less warpage. The greater the pressure variations, the greater the warpage is likely to be.

NOTE: Not all values displayed in the Hold pressure result occur at the same time, maximum pressure for each area can occur at different times during the packing phrase.

To reduce variation in the Hold pressure result, you can do the following:

- increase the packing pressure
- increase the packing time
- change the gate size
- change the packing profile

Things to look for

- Isolated areas of low pressure surrounded by areas of high pressure may occur where thick regions of the part are being fed from thin areas of the part. You may need to change the gate location, or part geometry.

- A uniform drop in the pressure as the flow front approaches the last areas to fill may indicate there is a problem with the gate size or packing profile. This usually happens when the gate is too small and freezes too early, or when packing pressure has not been applied for a long enough period. Increase the gate size or change the packing profile.

In-cavity residual stress in first principal direction result

25

The In-cavity residual stress in first principal direction result shows the stresses in the orientation direction before ejection.

This result is generated by a Pack analysis using Midplane or Dual Domain analysis technologies.

Using this result

Residual stresses in the part can be created as a result of shear stresses generated during mold filling or packing. In addition to these flow-induced stresses, residual stresses can also be created by different areas of the part cooling at differing rates due to variations in the part surface temperatures when it is ejected. To minimize these stresses, uniform cooling is required.

These residual stresses can be the cause of premature part failure in service or of part warpage and distortion.

NOTE: The in-cavity residual stress is calculated by the Pack analysis and represents the stresses in the part before it is ejected. It does not necessarily reflect the stresses that a part experiences after ejection. This result is best used as an input to Warp or Stress analyses, such as Warp or Abaqus, rather than by itself.

Things to look for

Positive values on the plot indicate tension and negative values indicate compression.

- In-cavity residual stresses are almost always positive because the part is still under constraint within the mold. The mold physically prevents the material from shrinking while the part is in the cavity. The result of that is a stress that keeps the element stretched in its place. However, when the part is ejected, this stress is relieved and the part may shrink.
- Negative values indicate overpacking has occurred.

For a more accurate indication of which portions of the part are under tension and which are under compression, use the Warp stress results.

In-cavity residual stress in second principal direction result

26

The In-cavity residual stress in second principal direction result shows the stresses before ejection in the direction perpendicular to the first principal direction.

This result is generated by a Pack analysis using Midplane or Dual Domain analysis technologies, and shows the stresses before ejection in the direction perpendicular to the first principal direction.

Using this result

Residual stresses in the part can be created as a result of shear stresses generated during mold filling or packing. In addition to these flow induced stresses, residual stresses can also be created by different areas of the part cooling at differing rates due to variations in the part surface temperatures when it is ejected. To minimize these stresses, uniform cooling is required.

These residual stresses can be the cause of premature part failure in service or of part warpage and distortion.

NOTE: The in-cavity residual stress is calculated by the Pack analysis and represents the stresses in the part before it is ejected. It does not necessarily reflect the stresses that a part experiences after ejection. This result is best used as an input to a Warp or Stress analyses, such as Warp or Abaqus, rather than by itself.

Things to look for

Positive values on the plot indicate tension and negative values indicate compression.

- In-cavity residual stresses are almost always positive because the part is still under constraint within the mold. The mold physically prevents the material from shrinking while the part is in the cavity. The result of that is a stress that keeps the element stretched in its place. However, when the part is ejected, this stress is relieved and the part may shrink.
- Negative values indicate overpacking has occurred.

For a more accurate indication of which portions of the part are under tension and which are under compression, use the Warp stress results.

Interface heat flux (3D Overmolding) result

27

The Interface heat flux result shows the amount of heat flowing from the current part into the previous part (i.e., from overmolding into original molding, or into a part insert). Negative values indicate heat flowing into the current molding.

This result is generated at the end of a Fill analysis using the Coupled 3D Flow solver.

Using this result

Only mesh faces which form the boundary between the current part and the previous part (or insert) are shown in this result. Blank areas indicate where the solver could not match mesh faces on one or the other part, and may indicate holes in the mesh interface.

NOTE: This result should not be relied upon to detect holes in interface meshing.

This result indicates how much heat is flowing between parts during insert or overmolding processes. Heat flow that is too high may cause remelting in the first part or insert. Heat flow that is uneven may cause stresses or warping in the first part.

Things to look for

Use this result in conjunction with the re-melt temperature result to check for areas which may have re-melted as a result of heat flow.

Use this result to check for possible warping in the previous part or insert.

Intermediate results 28

Intermediate results provide more time-animation capabilities when viewing results because intermediate time-slices have been saved.

Profiled Intermediate results allow you to view a variable, for example, temperature or velocity, across the cavity thickness at selected locations in the part and at a specified time from the start of injection.

You can adjust the number of intermediate results for each phase of the injection cycle. This allows you to obtain more detailed information about important parts of the analysis. Warpage results, for example, could benefit from more intermediate results through the packing phase.

By default in a Fill+Pack analysis using the Coupled 3D Flow solver, there are 5 intermediate steps in the filling phase, 5 intermediate steps in the packing phase and 3 intermediate steps in the cooling phase.

By default in a Midplane or Dual Domain Fill+Pack analysis there are 5 intermediate steps in the filling phase and 5 intermediate steps in the remainder of the analysis.

NOTE: If the injection time is set to Automatic, the intermediate results may not be equally spaced.

Red Pressure profile, **Green** Filling phase, **Brown** Packing phase, **Blue** Cooling phase.

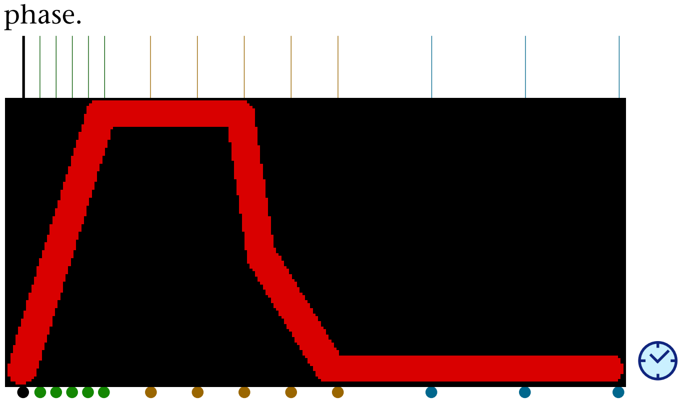


Figure 1: Intermediate results

Machine setup result

29

The Machine setup result is generated from a Fill analysis and shows the parameters that you should use on the injection molding machine, as determined by the analysis.

TIP: Right-click on the analysis log tab and select **Save As** to save the information to a text file.

30 Orientation at core result

The Orientation at core result provides a good indication of how molecules will be oriented at the part core, showing the average principal alignment direction for the whole element.

This result is generated from a Fill analysis using Midplane or Dual Domain analysis technologies,

Using this result

The core orientation for each triangular element is perpendicular to the velocity vector before the center layer reaches the transition temperature. This is the most probable orientation in the core region of a part. The other possible orientation is in the direction of the velocity vector.

Without rigorous Fiber orientation analysis, core orientation provides a good indication of how molecules or fibers will be oriented when using a fiber-filled material. The magnitudes of these vectors are normalized to one and are displayed multiplied by the given scale factor. Core orientation is in the transverse direction of the flow.

The linear shrinkage of a part also depends on the orientation. For unfilled polymers, the shrinkage in the direction of skin (flow) orientation is greater than in the direction of core (transverse) orientation. However, this situation may be reversed when using fiber-filled polymers, because of the low shrinkage and stiffness of the fibers in the direction of skin orientation.

Things to look for

- Check the core orientation direction. If the fibers are not oriented correctly, you may need to run a Fiber orientation analysis.
- Check the Orientation at skin result.

31 Orientation at skin result

The Orientation at skin result provides a good indication of how molecules will be oriented on the outside of the part, showing the average principal alignment direction for the whole local area at the end of filling.

Without rigorous Fiber orientation analysis, skin and core orientations provide a good indication of how molecules will be aligned in non-fiber filled materials. This result shows how fibers will be oriented at the skin when using a fiber-filled material.

NOTE: Core fiber orientation may be different to skin fiber orientation.

Because the melt freezes very quickly when it contacts the mold for the first time, the velocity vector provides the most probable molecular orientation at the skin.

The magnitudes of these vectors are normalized to one and are displayed multiplied by the given scale factor. Skin orientation is determined by the velocity direction when the melt front first reaches a given location.

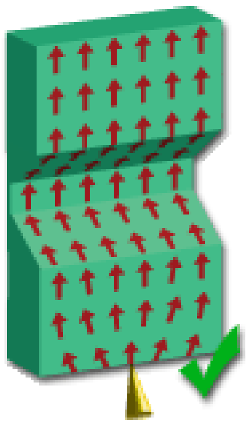
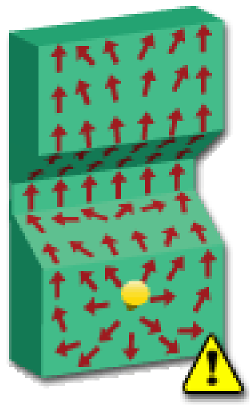
Using this result

The Orientation at skin result is useful for estimating the mechanical properties of a part. For example, the impact strength is typically much higher in the direction of molecular orientation at the skin. When using fiber-filled polymers, the tensile strength is also higher in the direction of orientation at the skin because the fibers on the surface are aligned in that direction. The skin orientation generally represents the direction of strength. For plastic parts that must withstand high impact or force, the gate location can be designed to give a skin orientation in the direction of the impact or force.

The linear shrinkage of a part also depends on the skin orientation. For unfilled polymers, the shrinkage in the direction of the skin or flow orientation is greater than in the direction of the core or transverse orientation. This situation may be reversed when using fiber-filled polymers due to the low shrinkage and stiffness of the fibers in the direction of skin orientation.

Compare the Orientation at skin result with the Fill time result to identify unbalanced flows in the part.

It is important to orientate the molecules correctly to ensure the mechanical quality of a hinge. The injection locations must be placed in positions that produce a consistent molecular orientation, as shown in the following diagram on the left. The diagram on the right shows inconsistent molecular orientation, which leads to lower part quality.

Things to look for

When viewing the Orientation at skin result, watch for the following:

- Unbalanced flow
- Inconsistent orientation

NOTE: More accurate predictions of fiber orientation can be obtained from a Fiber analysis.

Orientation at bottom skin result 32

The Orientation at bottom skin result is generated at the end of a Fill analysis using Midplane analysis technology, and shows the flow direction at the plastic/metal interface for the bottom side when plastic fills an element.

Without rigorous fiber orientation analysis, skin and core orientations provide a good indication of how molecules will be aligned in non-fiber filled materials. This result shows how fibers will be oriented at the skin when using a fiber-filled material.

NOTE: Core fiber orientation may be different to skin fiber orientation.

Because the melt freezes very quickly when it contacts the mold for the first time, the velocity vector provides the most probable molecular orientation at the skin.

The magnitudes of these vectors are normalized to one and are displayed multiplied by the given scale factor. Skin orientation is determined by the velocity direction when the melt front first reaches a given location.

Using this result

The skin orientation results provide a good indication of how molecules or fibers will be oriented when using a fiber-filled material, without performing a fiber orientation analysis.

Skin orientation is useful for estimating the mechanical properties of a part. For example, the impact strength is typically much higher in the direction of skin orientation. When using fiber-filled polymers, the tensile strength is also higher in the direction of skin orientation, because the fibers on the surface are aligned in that direction. Skin orientation generally represents the direction of strength. For plastic parts that must withstand high impact or force, the gate location can be designed to give a skin orientation in the direction of the impact or force.

The linear shrinkage of a part also depends on the skin orientation. For unfilled polymers, the shrinkage in the direction of skin (flow) orientation is greater than in the direction of core (transverse) orientation. However, this situation may be reversed when using fiber-filled polymers, because of the low shrinkage and stiffness of the fibers in the direction of skin orientation.

There are exceptions to the fiber orientation described above. For example, in the area of radial flow near a gate, polymer melt stretches in the transverse direction. Fibers are then more likely to be oriented in the transverse direction in both the skin and core.

Things to look for

- Unbalanced flow.

- Inconsistent orientation.

NOTE: More accurate predictions of fiber orientation can be obtained from a Fiber analysis.

Orientation at top skin result 33

The Orientation at top skin result is generated at the end of a Fill analysis using Midplane analysis technology, and shows the flow direction at the plastic/metal interface for the top side when plastic fills an element.

Without rigorous Fiber orientation analysis, skin and core orientations provide a good indication of how molecules will be aligned in non-fiber filled materials. This result shows how fibers will be oriented at the skin when using a fiber-filled material.

NOTE: Core fiber orientation may be different to skin fiber orientation.

Because the melt freezes very quickly when it contacts the mold for the first time, the velocity vector provides the most probable molecular orientation at the skin.

The magnitudes of these vectors are normalized to one and are displayed multiplied by the given scale factor. Skin orientation is determined by the velocity direction when the melt front first reaches a given location.

Using this result

The skin orientation results provide a good indication of how molecules or fibers will be oriented when using a fiber-filled material, without performing a fiber orientation analysis.

Skin orientation is useful for estimating the mechanical properties of a part. For example, the impact strength is typically much higher in the direction of skin orientation. When using fiber-filled polymers, the tensile strength is also higher in the direction of skin orientation, because the fibers on the surface are aligned in that direction. Skin orientation generally represents the direction of strength. For plastic parts that must withstand high impact or force, the gate location can be designed to give a skin orientation in the direction of the impact or force.

The linear shrinkage of a part also depends on the skin orientation. For unfilled polymers, the shrinkage in the direction of skin (flow) orientation is greater than in the direction of core (transverse) orientation. However, this situation may be reversed when using fiber-filled polymers, because of the low shrinkage and stiffness of the fibers in the direction of skin orientation.

There are exceptions to the fiber orientation described above. For example, in the area of radial flow near a gate, polymer melt stretches in the transverse direction. Fibers are then more likely to be oriented in the transverse direction in both the skin and core.

Things to look for

- Unbalanced flow.

- Inconsistent orientation.

NOTE: More accurate predictions of fiber orientation can be obtained from a Fiber analysis.

Percentile values in Fill analysis results

34

Percentiles are the set of numbers from 0 to 100 that divide a set of ranked data into 100 class intervals with each interval containing 1/100 of the observations.

A particular percentile, say the 5th percentile, is a cut point with 5% of the observations below it and the remaining 95% of the observations above it. Thus for an X percentile, at least X% of the observations are below this value.

Percentile results are available in the analysis log for a Fill analysis.

For example, in the **Filling phase results summary for the part**:

- Bulk temperature—95th percentile, suggests that at least 95% of the part volume is below or at this temperature.
- Bulk temperature—5th percentile, suggests that at least 5% of the part volume is below or at this temperature.

Using this result—95th and 5th Percentiles

The maximum and minimum values of variables are well defined. However, the 95th and 5th percentile values are more useful in the development of molding windows and for comparing results. The 95th and 5th percentile values are the relative position (weighted by volume) in an ordering of the values of a variable. In other words, 95% of the part or runner volume has a value of the variable below the 95th percentile value. Similarly, only 5% of the part or runner volume has a value of the variable below the 5th percentile value.

The 95th and 5th percentile values reflect the overall range of variation of the variables for the majority of the part or runners. The absolute maximum and minimum values may sometimes be misleading. For example, a few small, thin ribs on an otherwise uniform and relatively thicker part will result in a very low minimum temperature at the end of the filling stage. Using the 5th percentile value, the colder area on the rib can be excluded. This provides a more realistic lower bound for the temperature on the part. Similarly, the 95th percentile value can exclude the effect of excessive viscous heating occurring only near the gate region, and provide a better upper bound of the temperature on the part. Since the 95th and 5th percentile values provide a better idea of the variation of the particular variable under consideration, these values should be used to evaluate design considerations and to compare results between different designs.

Polymer fill region result (3D) 35

The Polymer fill region result is generated at the end of a Fill or Overmolding analysis using 3D analysis technology, and shows which elements in the model are filled with polymer melt at intervals as the cavity fills.

Filled elements are shown as green and unfilled elements as transparent.

Using this result

The Polymer fill region result is able to show where jetting flow has occurred. If the jetting defect has occurred, a region may fill and unfill repeatedly. This effect is not visible in the Fill time result because the Fill time result can show only one time (the first) that a node fills.

Jetting may result in uneven filling and may lead to air traps inside the part. It can also lead to uneven cooling and increased stress in the part.

Things to look for

Play this result as an animation to spot jetting, which will appear as streams of melted material twisting around.

TIP: Use the **Cutting Plane** tool to see an obscured jet more clearly.

Pressure result

36

The Pressure result is generated from a Fill analysis, and shows the pressure distribution through the flow path inside the mold at the time the result was written.

Pressure results derivation discusses the color distribution of this result. At the beginning of filling, the pressure is zero (or 1 atm, in the absolute pressure scale) everywhere in the mold. The pressure at a specific location starts to increase only after the melt front reaches that location. The pressure continues to increase as the melt front moves past, due to the increasing flow length between this specific location and the melt front.

The pressure difference from one location to another is the force that pushes the polymer melt to flow during filling. The pressure gradient is the pressure difference divided by the distance between two locations. Polymer always moves in the direction of the negative pressure gradient, from higher pressure to lower pressure. (This is analogous to water flowing from higher elevations to lower elevations). Thus, the maximum pressure always occurs at the polymer injection locations and the minimum pressure occurs at the melt front during the filling stage.

The magnitude of the pressure (or pressure gradient) depends on the resistance of the polymer in the mold, because polymer with high viscosity requires more pressure to fill the cavity. Restricted areas in the mold, such as thin sections or small runners, and long flow lengths also require a larger pressure gradient and thus higher pressure to fill.

NOTE: The Pressure result is an intermediate result, meaning its animation by default is through time and the scale by default is the minimum to maximum of the entire range of the result.

Using this result

Normally the maximum injection pressure at the nozzle is about 140 MPa (20,000 psi). We recommend having a maximum pressure of 100 MPa (14,500 psi) for the mold (part and feed system) and 70 MPa (10,000 psi) maximum for the part. There are many molding machines with higher pressure capacities. If you don't know what the pressure capacity is, assume it is 140 MPa.

If the pressure capacity of the molding machine is known, use no more than about 75% of the pressure capacity for a design guide for the entire mold, and 50% for just the part.

The default molding machine for the simulation has a 180 MPa pressure limit. You can use the pressure limitations two ways. Assume the injection molding for making the parts has a pressure limit of 140 MPa. You can run an analysis with the default 180 MPa limit and review the pressure requirements of the mold. It should not be over about 100 MPa, as a design guide line. If the analysis pressure is 120 MPa, the pressure should be reduced.

You could also set the molding machine limit to 100 MPa and run the analysis. You would be simulating a pressure limited process. The filling phase should be controlled by velocity, not pressure. Therefore the process should be fixed.

NOTE: The typical maximum hydraulic pressure of an injection molding machine ram is around 14 MPa. When the polymer is injected and is forced into the nozzle, there is a pressure intensification factor of between 8 -15 due to the smaller area of the ram compared to the hydraulic cylinder moving the ram. Therefore, the pressure available at the nozzle is normally between 110 MPa and 210 MPa. 140MPa is around the average.

Things to look for

- Pressure should be zero at the extremities of each flow path at the end of filling.
- Generally, the maximum pressure should be less than 100 MPa for parts with runners and 70 MPa for parts without runners.

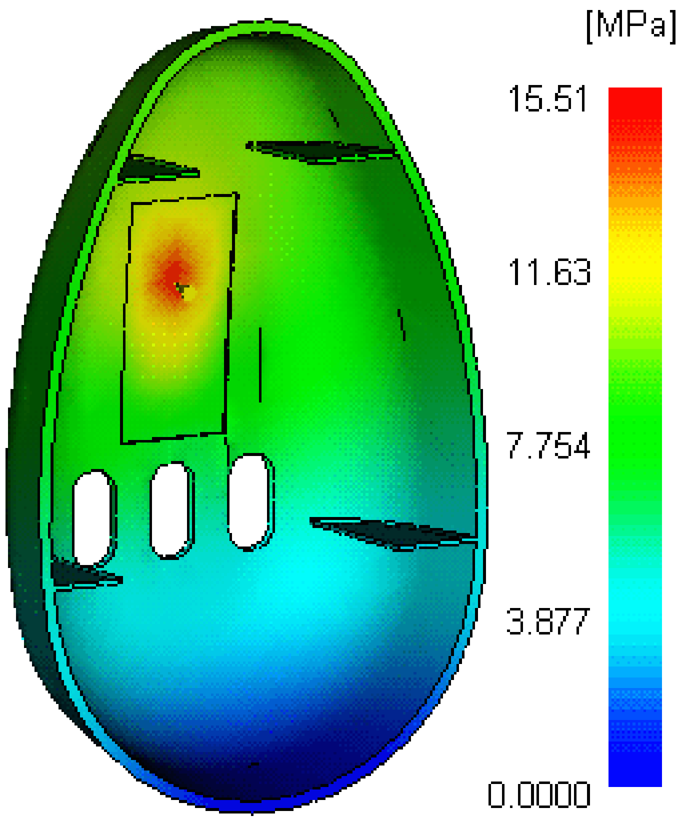
37 Pressure at end of fill result

The Pressure at end of fill result, which is produced by a Fill analysis, shows the pressure distribution in the cavity at the instant when the cavity is completely filled with polymer.

At the beginning of filling, the pressure is zero, or 1 atm in the absolute pressure scale, throughout the mold. The pressure at a specific location starts to increase only after the melt front reaches that location. The pressure continues to increase as the melt front moves past, due to the increasing flow length between this specific location and the melt front.

The pressure difference from one location to another is the force that pushes the polymer melt to flow during filling. The pressure gradient is the pressure difference divided by the distance between two locations.

Like water flowing from higher elevations to lower elevations, polymer always moves in the direction of the negative pressure gradient, from higher pressure to lower pressure; therefore, the maximum pressure occurs at the polymer injection locations and, the minimum pressure occurs at the melt front during the filling stage, as shown in the following diagram.



The magnitude of the pressure or pressure gradient depends on the resistance of the polymer in the mold. This is because polymer with high viscosity requires more pressure to fill the cavity. Restricted areas in the mold, such as thin sections, small runners, and long flow lengths, also require a larger pressure gradient, and, therefore, a higher pressure to fill.

Using this result

During the filling stage, large variations in the pressure distribution, which are indicated by closely-spaced contours, should be avoided. Pressure should be zero at the extremities of each flow path at the end of filling.

During packing, pressure variations affect the volumetric shrinkage.

The pressure variation in the cavity should also be minimized during the packing stage.

NOTE: The typical maximum hydraulic pressure of an injection molding machine ram is around 20 MPa. When the polymer is injected and is forced into the nozzle, there is a pressure intensification factor of between 8 -15 due to the smaller area of the nozzle. Therefore, the pressure available at the nozzle is normally between 160 MPa and 300 MPa. 200 MPa is around the average.

Pressure at injection location result 38

The Pressure at injection location result shows the pressure at the injection location at various times during the filling and packing phases of the analysis.

Using this result

All of the injection locations on the model will have the same pressure. This is an assumption used in the analysis.

The Pressure at injection location result is very useful for checking whether there are any pressure spikes, which is normally a sign of imbalance. This could be within a part or between parts. If it is within a part, normally this could be fixed by changing gate location(s). Sometimes only a subtle change is necessary.

The Pressure at injection location result is an XY plot generated from Fill and Microchip Encapsulation analyses.

Things to look for

- A significant change in the slope of the pressure curve, during the filling phase, indicates a change in the pressure gradient in the part. Often this is caused by a flow imbalance within a part or for multi-cavity models between parts. Review the fill time result and other pressure results for balance problems.
- The packing profile is shown with this result. The default packing profile uses 80% of the pressure at switch over for 10 seconds. On this result, the pressure will drop instantaneously 20% then the line will be horizontal for 10 seconds.
- This result can also be used to ensure the packing profile is correct.

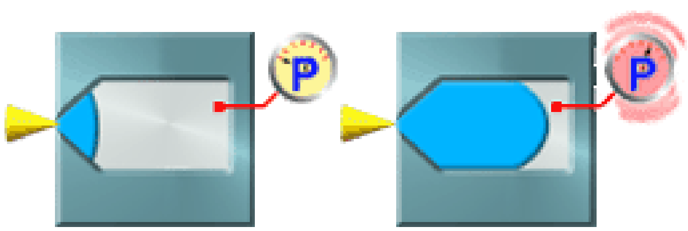
39 Pressure at velocity/pressure switchover result

The Pressure at V/P switchover result is generated from a Fill analysis, and shows the pressure distribution through the flow path inside the mold at the switchover point from velocity to pressure control.

Using this result

Pressure should be zero at the extremities of each flow path at the end of filling.

NOTE: The typical maximum hydraulic pressure of an injection molding machine ram is around 14 MPa. When the polymer is injected and is forced into the nozzle, there is a pressure intensification factor of between 8 -15 due to the smaller area of the nozzle. Therefore, the pressure available at the nozzle is normally between 110 MPa and 210 MPa. 140 MPa is around the average.



Things to look for

During the filling stage, large variations in the pressure distribution, indicated by closely-spaced contours, should be avoided.

Normally the maximum injection pressure at the nozzle is about 140 MPa (20,000 psi). We recommend having a maximum pressure of 100 MPa (14,500 psi) for the mold (part and feed system) and 70 MPa (10,000 psi) maximum for the part. There are many molding machines with higher pressure capacities. If you don't know what the pressure capacity is, assume it is 140 MPa. If the pressure capacity of the molding machine is known, use no more than about 75% of the pressure capacity for a design guide for the entire mold, and 50% for just the part.

There will be zero pressure at the flow front, and some portion of the part is not filled. The portion not filled as shown in this plot will be filled under pressure control as defined in the process settings.

40 Re-melt zone, overmolded components (3D Overmolding) result

The Re-melt zone, overmolded components (3D Overmolding) result shows places where the first component may not have completely frozen, or has re-melted, when the second component is injected.

This result is generated at the end of an Overmolding Fill analysis using 3D analysis technology.

You can control how re-melt zones are displayed by editing the plot properties.

Using this result

Re-melting occurs when the first component has not cooled enough before the second component is injected, or when the melt temperature of the second component is sufficient to heat the material in the first component above its transition temperature. This result tells you where, on the boundary between the first and second components, re-melting is likely to occur.

Re-melting of a thin skin increases structural strength between the components. However, it is generally undesirable because it causes unpredictable changes to the properties of the first component, such as its exact shape or optical properties.

Things to look for

To prevent excessive re-melting, try one of these options:

- Increase the cycle time, particularly between injection of the first and second components.
- Decrease the injection temperature of the material in the second component.
- Change the material for either component.
- Move the second injection location to be further away from the components' boundary.

Re-melt zone, part insert result 41

The Re-melt zone, part insert result shows the location(s) where the insert may have melted when the part is injected.

This result is generated at the end of a Fill analysis using 3D analysis technology, where an insert is modeled.

You can control how re-melt zones are displayed by editing the plot properties.

Using this result

Re-melting occurs when the insert material temperature increases above its transition temperature because of heat transfer from the part. This result tells you where, on the boundary between the part and the insert, re-melting is likely to occur.

Re-melting of a thin skin increases structural strength between components. However, it is generally undesirable because it causes unpredictable changes to the properties of the insert, such as its exact shape or optical properties.

Things to look for

To prevent excessive re-melting, try one of these options:

- Decrease the injection temperature of the material.
- Change the insert or part material.
- Move the injection location to be further from the insert.

42 Ram speed, recommended result

The Ram speed, recommended result defines an injection profile that keeps the flow front velocity constant.

Using this result

The warpage of a part is often related to variations of the flow front velocity during the filling stage of the analysis. The higher the velocity in the fountain (flow front) region, the higher the surface stress and the degree of molecular orientation.

The Ram speed, recommended result, displayed as an XY plot, is used to maintain a constant flow front velocity throughout filling. The ram speed is pro-rated by the flow front area calculated at every instant: the larger the flow front area, the higher the ram speed needed to maintain a constant flow front velocity.

Stress is related to the pressure drop. For a given cross-sectional area, a higher flow rate corresponds to a higher pressure drop, and therefore, a higher stress. To minimize stress, it is desirable to have a slower flow rate through smaller cross-sections, while it is possible to have a faster flow rate through larger cross-sections. Maintaining a constant flow front velocity by changing the ram speed of a closed-loop process controller helps to minimize variations in surface stress during the process, and thus decreases the tendency of the part to warp.

Things to look for

- Use the optimal injection profile indicated in this result to maintain a constant flow front velocity in the mold.

TIP: The Ram speed, recommended result is also listed in the analysis log file. This profile can be directly entered in the Relative ram speed profile %flow rate vs. %shot volume.

Shear rate result

43

The Shear rate result shows the rate of shear strain (the velocity gradient through the cross-section) in the mold-cavity at the time the result was written.

In 3D, this result takes into account shear-induced imbalance in a multi-cavity layout.

This is an intermediate result which is available for all analysis technologies.

Using this result

The shear rate is a measure of how quickly the layers of plastic are sliding past each other. If this happens too fast, the polymer chains break and the material degrades.

When the Shear rate result is displayed, scale the result using **Results tab > Properties panel > Plot Properties** and select the **Scaling** tab to show very high shear rates. The bulk shear rate should be less than the maximum recommended for the material.

The Shear rate result may have small portions of the cross–section that are above the the maximum material limit while the bulk shear rate remains below this limit. Regions above the limit could be subject to material degradation, embrittlement and poor surface finish.

When the maximum shear rate is much larger than the bulk shear rate, it is useful to look at the shear rate as an XY graph.

Things to look for

- High shear rates tend to occur in the feed system (where the greatest velocities are).
- To reduce the shear rate, increase the cross–section or decrease the flow rate

Shear rate result

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

Finding the location of maximum shear rate

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

   The textual log files appear below the model pane in the logs pane.

3. Select the **Analysis Log** tab in the logs pane.

   A textual description of the Analysis Log appears.
4. Scroll the Analysis Log to locate the section titled “**Filling phase results summary for the part**”.
5. Note the time at which the **Shear rate—maximum value** occurs.

   The time value appears in the parenthesis. For example: `Shear rate—maximum (at 3.407 s).`
6. Display the **Shear rate, bulk** result.
7. Click **Step Forward** repeatedly until the result frame corresponds to the time you noted earlier.

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

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

Shear rate result (3D) 44

The Shear rate (3D) result shows the shear rate in the part and the feed system at the time the result was written.

This result relates to the guideline provided in the material database and is generated from a Fill analysis using 3D analysis technology.

Using this result

The shear rate is a measure of how quickly the layers of plastic are sliding past each other. If this happens too fast, the polymer chains break and the material degrades.

When the Shear rate (3D) result is displayed, scale the result using **Results tab > Properties panel > Plot Properties** and select the **Scaling** tab to show very high shear rates. The shear rate should be less than the maximum recommended for the material.

The shear rate result may have only small portions of the cross–section that are above the maximum material limit. Regions above the limit could be subject to material degradation, embrittlement and poor surface finish.

When the maximum shear rate is near or above the shear rate limit, it is useful to look at the shear rate as an XY graph.

TIP: Use cutting planes to view shear rate results through a solid model.

Things to look for

- High shear rates tend to occur in the feed system (where the greatest velocities are).
- To reduce the shear rate, increase the cross–section or decrease the flow rate

Shear rate, bulk result

45

The Shear rate, bulk result plot shows the magnitude of the shear rate through a cross-section.

Shear rate, bulk is derived from the wall-shear stress and the fluidity, and characterizes the magnitude of the shear rate through a cross-section. A representative viscosity is first calculated from the fluidity and the thickness of the part. The Shear rate, bulk is then calculated from the wall-shear stress and this representative viscosity.

NOTE: The Shear rate, bulk result is an intermediate result, meaning its animation by default is through time and the scale by default is the minimum to maximum of the entire range of the result.

Using this result

The shear rate is a measure of how quickly the layers of plastic are sliding past each other. If this happens too fast, the polymer chains break and the material degrades.

The bulk shear rate should not exceed the maximum value recommended for the material in the material database. Exceeding this value would likely lead to polymer degradation.

As temperature does, shear rate varies with thickness. A bulk shear rate gives an overview of the shear rate distribution in the filling stage. In contrast to the bulk temperature, the bulk shear rate is not an average or weighted average of the shear rate across the thickness. An average or weighted average is not suitable because the shear rate can vary widely across the part thickness.

Things to look for

- High shear rates tend to occur in the feed system (where the greatest velocities are).
- To reduce the shear rate, increase the cross–section or decrease the flow rate.

Shear rate, maximum result 46

The Shear rate, maximum result shows the maximum shear rate at a given node, up to the time the result was written.

For example, if the shear rate at a given node reaches a maximum of 4200 1/s at time = 1.5 seconds, this is the value given for all subsequent times.

This result is generated from a Fill analysis using 3D analysis technology.

Using this result

The shear rate is a measure of how quickly the layers of plastic are sliding past each other. If this happens too fast, the polymer chains break and the material degrades.

When the Shear rate result is displayed, scale the result using **Results tab > Properties panel > Plot Properties** and select the **Scaling** tab to show very high shear rates. The bulk shear rate should be less than the maximum recommended for the material.

The Shear rate result may have small portions of the cross–section that are above the the maximum material limit while the bulk shear rate remains below this limit. Regions above the limit could be subject to material degradation, embrittlement and poor surface finish.

When the maximum shear rate is much larger than the bulk shear rate, it is useful to look at the shear rate as an XY graph.

NOTE: Use cutting planes with this result to check the maximum shear rate that is occurring inside the solid model.

Things to look for

- High shear rates tend to occur in the feed system (where the greatest velocities are).
- To reduce the shear rate, increase the cross–section or decrease the flow rate

To reduce internal shear strain:

- Slow down the flow rate.
- Use programmed injection speeds.
- Increase wall thickness.
- Avoid differential orientation and differential shrinkage.

Shear stress at wall result 47

The Shear stress at wall result shows the shear stress at the plastic frozen/molten interface at the time the result was written.

This result is an intermediate result, meaning there are 20 results written by default during fill, and relates to the guideline provided in the material database.

The Shear stress at wall result is generated from a Fill analysis using Midplane or Dual Domain analysis technology.

Using this result

While shear stress is not the actual residual stress in a part, it is related to it. It is a measure of the factors effecting the degree of orientation of the melt next to the frozen layer. Orientated materials tend to shrink more than unoriented materials, so a large amount of orientation near the edge of the melt compared to near the center will lead to higher residual stress. Higher residual stresses can result in parts stress–cracking during ejection or in service.

Wall-shear stress is the shear force at the frozen/molten interface, per unit area, and is proportional to the pressure gradient at each location (If the polymer cross-section is completely molten, the frozen/molten interface is at the mold wall). Under the viscous flow formulation, shear stress is zero at the center of the cross–section and linearly increases to the frozen/molten interface. Thus, the wall-shear stress could be at its maximum value at any portion of the cross-section.

The shear stress should be less than the maximum recommended for the material in the material database. The shear stress can be compared directly with the values stored in the material database. Regions above this limit could be subject due to stress-cracking during ejection or in service.

NOTE: Thermoset materials do not have maximum recommended shear stress values in the material database.

Shear stress is also an indirect indication of the degree of molecular or fiber orientation. Higher shear stress would induce higher orientation, especially near the surface of the part. A more precise prediction of fiber orientation can be obtained from a Fiber analysis.

TIP: To determine where and when high stress occurs, you can set the scale with the minimum value to equal the shear stress limit, set the part to transparent, and deselect **Nodal averaged** in the **Plot Properties—Optional Settings** page. Only elements with a high shear stress are displayed.

Things to look for

- Check the wall shear stress is less than that recommended for the material.

  The shear stress in the part is important, but not in the feed system. Only when additives to the polymer are very shear sensitive does the shear stress in the gates and runners become important. The material shear stress limit can be found in the Material Database.

- Local thickening at the location of the high shear stress can be used to reduce shear stress.
- Slower flow rates may reduce the maximum shear stress.

  When the flow rate is at or near the end of fill, use an injection profile or switch to pressure control earlier to reduce the flow rate as the problem area is filling. If the high stress is near the gate or somewhere in the middle of the part, a longer injection time may reduce the shear stress. Use care when increasing the injection time as the melt temperature in the part may drop too much causing other filling and packing problems.

- Changing the material to a less viscous material or increasing the melt temperature will also reduce shear stress.

Sink marks, index result

48

The Sink marks, index is an indication of the potential shrinkage due to a hot core.

It is calculated for each element at the instant when local pressure has decayed to zero during the packing stage, and reflects how much material is still melt and left unpacked. Higher Sink marks, index value shows higher potential shrinkage. However, whether or not the shrinkage would result in sink mark depends on geometry characteristics.

The Sink marks, index generated indicates the likely presence and location of sink marks (and voids) in the part.

Using this result

The Sink marks, index indicates a degree of severity of depth as affected by the material, part geometry, position relative to the injection location and mold filling conditions. Changing any of these would allow you to determine its contribution to the sink marks severity. Generally, if the thickness of a rib is less than or equal to 60% of the main wall section, then there are likely to be no significant sink marks.

When the local pressure has decayed to zero during the packing and cooling stage, the Sink marks, index for the part is calculated as follows:

S=x+−x−⊠trans⊠atm−∫x−x+⊠dz2⊠⊠trans⊠atm

Where:

T_{trans}	is the transition temperature of the polymer
x^+	is the upper interfacial location where the temperature of the polymer is at the b5 value in the 2-domain Tait pvT model
x^-	is the lower interfacial location
⊠⊠	is the solid density of the polymer
p_{atm}	is the atmospheric pressure
h	is the half-gap thickness

The terms of the Sink marks, index equation can be interpreted as

((ideal mass of the liquid polymer) – (actual mass of the liquid polymer))/(ideal mass of the total thickness)

where the ideal mass is determined from the solid density value in the material properties, and the actual mass is determined from the pvT data.

The integration term in the Sink marks, index equation adds the density of the layers at the point where they become frozen. If the pressure at that point is high, then it is possible for the calculated total actual mass to be higher than the ideal packed density resulting in a negative Sink marks, index value. Negative Sink marks, index values therefore indicate overpacking in that area of the part. If the Sink marks, index is high, then it is possibly because a significant portion of the melt is freezing under zero, or low pressure. The larger the volume that freezes under low pressure, the higher the Sink marks, index and the greater the likelihood of a sink mark.

If it is not possible to remove or reduce a sink mark to the design criteria, you can conceal it. This can be done by adding a design feature, such as a series of serrations on the area where they occur.

What to do next

After identifying sink marks on your model, consider the following actions to remove them:

- Alter part design to avoid thick sections and reduce the thickness of any extrusions, or conceal the sink marks within the design.
- Increase the packing pressure/packing time.
- Relocate the gate to or near a thicker section. This allows these sections to be packed before the thinner sections freeze.
- Increase the size of gates and runners to delay the gate freeze-off time. This will allow more material to be packed into the cavity.
- Decrease the melt and mold temperature.
- Use a material with a lower viscosity.

Sink marks, depth result 49

The Sink marks, depth result indicates the location and depth of sink marks (and voids) likely to be caused by features on the opposite face of the surface.

Sink marks can occur in moldings with thicker sections, or at locations opposite ribs, bosses or internal fillets. The Sink marks, depth result only indicates sink marks likely to be caused by features on the opposite face of the surface.

The Sink marks, depth result plot is generated by a Pack analysis using Midplane analysis technology.

Using this result

Sink marks appear as depressions on the surface of a molded part. These depressions are typically very small; however they are often quite visible, because they reflect light in different directions to the part. The visibility of sink marks is a function of the color of the part as well as its surface texture so depth is only one criterion. Although sink marks do not affect part strength or function, they are perceived to be severe quality defects.

Sink marks are caused mainly by thermal contraction (shrinkage) during cooling. For example, areas where a rib meets the top surface will cool slower than other regions due to restricted local contact with the colder mold surface. This in turn will cause higher shrinkage which effectively pulls the cooling plastic away from the mold surface to create depressions on the surface of a molded part.

What to do next

After identifying sink marks on your model, consider the following actions to remove them:

- Alter part design to avoid thick sections and reduce the thickness of any extrusions, or conceal the sink marks within the design.
- Increase the packing pressure/packing time.
- Relocate the gate to or near a thicker section. This allows these sections to be packed before the thinner sections freeze.
- Increase the size of gates and runners to delay the gate freeze-off time. This will allow more material to be packed into the cavity.
- Decrease the melt and mold temperature.
- Use a material with a lower viscosity.

Sink marks estimate result 50

The Sink marks estimate result displays simulated sink marks on the part.

The Sink marks estimate result displays the calculated depths of sink marks in the part, and shows a legend to detail the depth differences.

This result indicates the presence and location of sink marks (and voids) likely to be caused by features on the opposite face of the surface. Sink marks typically occur in moldings with thicker sections, or at locations opposite ribs, bosses or internal fillets. The result does not indicate sink marks caused by locally thick regions.

NOTE: You can modify the properties of the Sink marks estimate result in order to *visualize a shaded result.*

Using this result

As sink marks are a visual rather than a structural defect, the result should be evaluated against the part's visual design specifications. Lighter colors and textured surfaces tend to make sink marks less visible.

The result index indicates a degree of severity of depth as affected by the material, part geometry, position relative to the injection location and mold filling conditions. Changing any of these would allow you to determine its contribution to the sink marks severity.

Generally, if the thickness of the rib is less than or equal to 60% of the main wall section then, there are likely to be no significant sink marks.

If it is not possible to remove or reduce a sink mark, you can conceal it. This can be done by adding a design feature, such as a series of serrations on the area where it occurs.

Things to look for

To reduce the occurrence of sink marks try one of these options:

1. Alter part design to avoid thick sections and reduce the thickness of any features that intersect with the main surface.
2. Relocate the gate to or near the problem areas. This allows these sections to be packed before the thinner sections between the gate and the problem areas freeze.
3. Increase the size of gates and runners to delay the gate freeze-off time. This will allow more material to be packed into the cavity.
4. Sometimes, decreasing the melt and mold temperature may be sufficient. Alternatively, a less viscous melt can be used.

Sink marks estimate result

Visualizing a sink marks shaded result

You can modify the properties of the Sink marks estimate result in order to visualize a shaded result.

1 Open an analysis with Sink marks results, or run a pack analysis.
2 Select the Sink marks estimate result.
3 Right-click on the result and then select **Properties**.
4 In the **Optional Settings** tab, select the **Single color** option from the **Color** group.

TIP: You can select a color that matches the color of your part.

---

5 Optionally, change the **Scale factor** value in the **Deflection** tab, to magnify the sink marks display.

51 Temperature result

The Temperature result shows the plastic temperature at a specified time, over all laminates through the part thickness.

This is a profiled time-series result which can be specified at constant intervals or at certain times.

Using this result

The Temperature display shows the plastic temperature results through the part thickness at a specified time.

NOTE: The plastic temperature at the mold-melt interface may differ from the specified mold surface temperature due to the heat transfer coefficient (HTC) settings.

If the flow front temperature is too low in a thin area of the part, hesitation or short shot may occur. In areas where the flow front temperature is too high, material degradation and surface defects may occur.

Make sure the flow front temperature is always within the recommended temperature range for the polymer you are using.

TIP:

- Keep the cooling and packing pressure distribution as uniform as possible to minimize warpage.
- Injection profiles may be required to obtain satisfactory temperature distribution.

Things to look for

- Hot spots, indicating excess shear heating usually around the last place to fill and around the gate. Try to acheive more uniform cooling.
- Cold spots, indicating hesitation.
- Excessive shear heating or cooling.

Temperature result (3D) 52

The Temperature result shows the temperature of the polymer at nodes, at specified times during the analysis. This result is generated by a Fill analysis using 3D analysis technology, and may be refined by a subsequent Pack analysis.

The Temperature display shows the plastic temperature results through the part thickness at a specified time. The plastic temperature at the mold-melt interface is the mold surface side of the interface due to the heat transfer coefficient (HTC) being used in the temperature calculations.

Using this result

The Temperature result can be used in many ways:

- Use a cutting plane to view the temperature gradient through the thickness.
- Animate the result to view the temperature variations over time.
- Use a single contour set at the transition temperature to determine when the gate has frozen.
- Use a single contour set at the ejection temperature to identify the areas that freeze last. These areas control the cycle time.
- Plot the temperature as a Probe XY plot to se a 2D graph of the temperature through the cross section.

NOTE: To increase the freeze time resolution, increase the number of intermediate files requested during the packing phase.

Thermoplastic 3D Overmolding

The mold and melt temperatures used in the overmolding component are initialized by the temperatures at the end of the first component stage. Therefore, the Temperature result shows the temperatures on the first component (first cavity), as well as the temperatures on the overmolding component (second cavity). This allows you to see the effect of the overmolding filling stage on the first cavity.

Things to look for

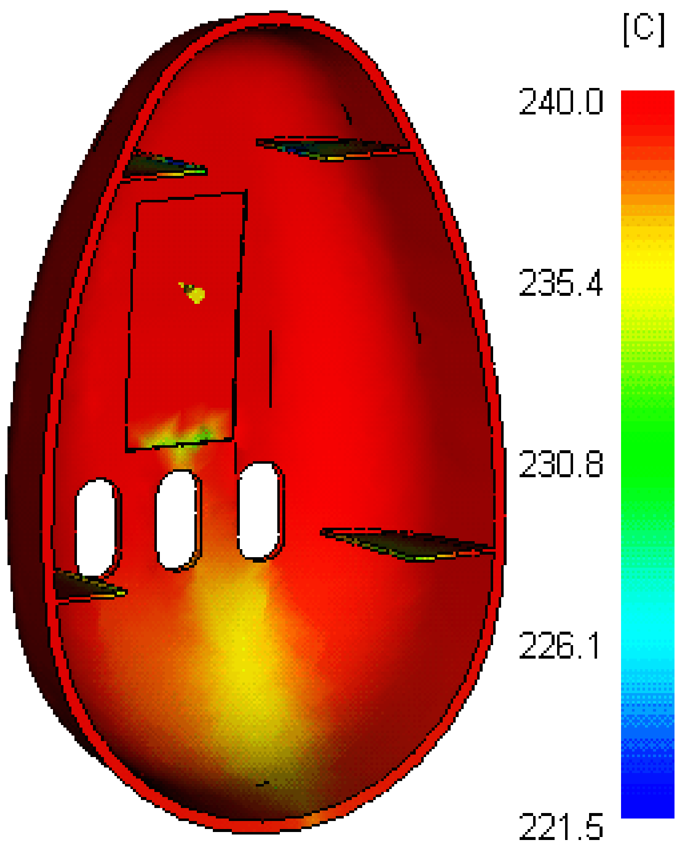
- Areas that freeze early, preventing the part from being adequately packed.
- Check when the gate has frozen, using a single contour, and noting where the contours seperate.
- Hot spots, indicating excess shear heating usually around the last place to fill and around the gate.

- Check mold cooling rates, and see whether there are hot or cold spots within the mold.
- Cold spots, indicating hesitation.

Temperature at flow front result 53

The Temperature at flow front result, which is produced by a Fill analysis, shows the temperature of the polymer when the flow front reaches a specified point in the center of the plastic cross-section.

As shown in the following diagram, the Temperature at flow front result uses a range of colors to indicate the region of lowest temperature in blue through to the region of highest temperature in red. The colors represent the material temperature at each point as that point was filled. The result shows the changes in the temperature of the flow front during filling.



The flow front temperature should not drop more than 2°C to 5°C during the filling phase. Larger changes often indicate that the injection time is too low, or there are areas of hesitation. If the flow front temperature is too low in a thin area of the part, hesitation may result in a short shot. In areas where the flow front temperature increases by several degrees, material degradation and surface defects may occur.

Using this result

Minimize the flow front temperature variation in the filling phase to under 2-5°C. Smaller temperature variations cause fewer problems.

- Filling profiles may be required to obtain satisfactory temperature distribution during the filling phase.

Try changing the following settings to improve the result, using the information in the table below to check whether a change has introduced other problems.

Problem	Change	Possible problems
Temperature too low	Decrease injection time.	May cause excess shear in the gate if it is restrictive. Too much shear will cause degradation and surface defects.
	Increase the melt temperature.	■ May increase cycle time. ■ May cause material degradation. Keep the temperature within the temperature limits specified in the material information in the database.
	Increase mold temperature.	May increase cycle time.
	Increase the thickness in the area to permit flow.	May cause a functional problem with the design and increase cost.
	Move the gate away from areas with hesitation.	May cause hesitation or other problems elsewhere in the part.
Temperature too high	Increase the injection time.	May cause hesitation.

Things to look for

When viewing the Temperature at flow front result, watch for the following:

- Hot spots; usually these exist around the gate and in areas where excessive shear heating occurs.
- Cold spots indicating hesitation.

- Check whether the material is shear heating or cooling excessively?

Tensile modulus, principal direction (fiber) results 54

The Tensile modulus, principle direction (fibre) result indicates how much stress is needed to cause a unit of movement.

This help topic describes the following fiber orientation Pack analysis results, and indicates how much stress is needed to cause a unit of movement:

- Tensile modulus in first principal direction (fiber)
- Tensile modulus in second principal direction (fiber)
- Tensile modulus in third principal direction (fiber)—3D analysis technology only

Using these results

For a Fiber and Dual Domain Fiber analysis, the Tensile modulus in first and second principal (fiber) results generate fiber-over-thickness (FOT) averaged values, which means that they are element-based values averaged over all the laminates of each element. For a 3D Fiber analysis, Tensile modulus, principal direction (fiber) results are recorded for each element in the model at the end of the analysis.

The first principal direction coincides with the fiber orientation first principal direction, and is determined by the Fiber orientation Fill+Pack analysis. The second principal direction is perpendicular to the first principal direction. If the first and second principal directions are X, and Y respectively, then Z is the third principal direction.

View these results in conjunction with one another to determine the average tensile moduli pressures (Mpa) in the first, second, and third principal directions.

NOTE: For more detailed information on the tensile modulus, view the Tensile modulus in first and second principal direction results, which are recorded for each laminate in the model using Midplane or Dual Domain technology.

Orthotropic assumption

The thermo-mechanical property calculation for fiber-filled composites are based on the orthotropic assumption, that fiber-filled material properties are different in three orthogonal principal directions. Under this assumption, there are 9 independent mechanical constants and three independent thermal expansion coefficients. In Midplane/Dual Domain models, because of the plain stress assumption in the shell structure analysis in warp, only 4 mechanical constants (Tensile modulus in first/second principal directions, Poisson ratio v12, Shear modulus G12) are necessary.

The **Orthotropic set** option selects the 9 mechanical constants (E1, E2, E3, v12, v23, v13, G12, G23, G13) and 3 CTE's (thermal expansion coefficient in first/second/third directions) all at once. In a Fiber analysis using 3D analysis technology, the complete set of thermo-mechanical properties with orthotropic assumption is necessary for a Warp analysis using 3D analysis technology. These properties are element-based, so each tetrahedral or beam element has its own orthotropic set of properties.

NOTE: To access the **Orthotropic set** option ensure you have selected an analysis sequence that includes Fill+Pack.

1 Click **Home tab > Molding Process Setup panel > Process Settings** . The **Process Settings Wizard** dialog opens.
2 If necessary, click **Next** until you reach the **Fill+Pack Settings** page of the Wizard.
3 Select the option **Fiber orientation analysis if fiber material**, and then click **Fiber parameters**. The **Fiber Orientation Solver Parameters** dialog opens
4 Click **Composite property calculation options**, and then from the **Fiber-filled property output** drop-down list, select **Orthotropic set**.

Throughput result

55

The Throughput result shows the total volume of material that passed through each beam element (runner system) connected directly to the injection node(s).

This result is generated from a Fill analysis using Midplane or Dual Domain analysis technology.

Using this result

The Throughput result is primarily used to check for balanced flow for a multi-gated or multi-cavity design.

The Throughput result will show you the volume of material that has passed through each beam element in the runner system. The volume should be largest at the sprue and smallest at the gate.

There are no Throughput results for triangular elements. Generally, there should be a similar volume of material passing through each gate of a multi-gated part.

Things to look for

- Unbalanced flow.

Time to reach ejection temperature result

56

The Time to reach ejection temperature result shows the amount of time required to reach the ejection temperature, which is measured from the start of fill.

If the part has not frozen by the end of the cycle time provided, a projected time to freeze is displayed in the result.

For a Midplane/Dual Domain analysis, the value displayed is the time taken for 100% of the local thickness to reach ejection temperature.

For 3D flow, time values are at the individual nodes. Use a cutting plane to investigate the (normally longer) internal values.

This result takes into account the dynamics of the packing phase, and where new hot material enters the cavity. This new hot material affects the cooling time.

NOTE: This result is not available for thermoset materials

Using this result

Ideally, the part should freeze uniformly. Areas of the part that take longer to freeze may indicate thicker areas of the part or areas of shear heat during filling and/or packing.

If a long period of time to reach the ejection temperature is caused by thick areas in the part, consider redesigning the part. Long periods of time that are due to shear may be difficult to solve. Reducing the shear may cause the Time to reach ejection temperature to adversely affect volumetric shrinkage and warpage.

If the resulting values for the part as a whole appear high, action to reduce the cycle time may need to be taken, such as reducing the mold and melt temperatures.

Things to look for

- Is the part sufficiently frozen at the end of the packing and cycle time?
- Is the gate freezing off too early before the cavity can be adequately packed out?
- Are there thinner regions freezing off before more extreme thicker regions thus preventing the thicker regions from being adequately packed out?

Unfilled cavity result

57

The Unfilled cavity result, which is produced by a Fill analysis using a 3D analysis technology, shows areas in a model where the part has not filled.

Unfilled areas are displayed in dark blue, which is the default color, and filled areas appear translucent. This result enables you to see short shots in the interior of the part, which may not be visible when viewing the Fill time result.

NOTE: This Unfilled cavity result is only available if a short shot exists in the model.

Using this result

This result should be used in conjunction with the Fill time result to identify areas of the cavity that have not filled.

Things to look for

Unfilled areas (short shots) are displayed in dark blue. Filled areas are translucent.

Velocity result

58

The Velocity result shows the magnitude of flow velocity in every laminate, at every element, at several times during the filling and packing phases.

The result can be animated over time up to when the file was written, animated across the part thickness at a specified time, and displayed as an XY plot over time or across part thickness.

NOTE: The Velocity result is an intermediate profiled result, meaning its animation by default is through time and the scale by default is the minimum to maximum of the entire range of the result.

Using this result

The velocity result can be used to determine areas with high or low flow rates. High velocity values for a section of a model indicates there is a high flow rate, meaning there could be shear heating problems. Very high local velocity values can cause the temperature to drastically increase in that area, resulting in surface blemishes, unbalanced filling, and packing and warpage problems.

If polymer flows quickly through one section of the mold, and flows slowly through another section of the mold, it could indicate filling defects such as hesitation or racetrack effects. Areas of the mold with very low velocities for most of the filling phase could indicate overpacking.

Things to look for

Areas of varying velocity, indicating:

- Overpacking.
- Hesitation.
- Racetrack Effect.
- Unbalanced Flow.

Velocity result (3D)

59

The Velocity result shows the direction and magnitude of velocity of the polymer, recorded at a node, when the result file was written.

The recorded value is the actual velocity of the polymer. This result is generated from a Fill analysis using 3D analysis technology,

Using this result

The default method of displaying the velocity as a vector, as a dart on a transparent model. Often the gate or another narrow cross section has a very high velocity. The velocity in the rest of the part is displayed at the low end of the scale, making it difficult to see differences. Use the plot properties to scale the results to exclude the high velocity seen in narrow cross sections. When you select the **Extended color** check box, areas with a velocity higher than the displayed maximum are red.

You can use the plot properties to plot the velocity in the X, Y or Z directions, and to specify the velocity dart length.

It is recommended that you use the Cutting Plane post-processing tool to view velocity results inside the solid model. The velocity result recorded for the surface of the model will be zero. You will need to first define the cutting plane, and then move it through the model.

The velocity result can be used to determine areas with high or low flow rates. High velocity values for a section of a model indicates there is a high flow rate, meaning there could be shear heating problems. Very high local velocity values can cause the temperature to drastically increase in that area, resulting in surface blemishes, unbalanced filling, and packing and warpage problems.

If polymer flows quickly through one section of the mold, and flows slowly through another section of the mold, it could indicate filling defects such as hesitation or racetrack effects. Areas of the mold with very low velocities for most of the filling phase could indicate overpacking.

Things to look for

Areas of varying velocity, indicating:

- Overpacking.
- Hesitation.
- Racetrack Effect.
- Unbalanced Flow.

Vent region pressure result 60

The Vent region pressure shows the predicted air pressure within air traps or at regions connected to specified venting analysis (air vent) locations.

This result is generated by a Fill or Fill+Pack analysis sequence for Thermoplastics Injection Molding, Thermoplastics Overmolding, Reactive Molding or Microchip Encapsulation molding processes using 3D analysis technology, when the option to **Perform venting analysis** is enabled.

NOTE: The Vent region pressure result is an intermediate result, meaning its animation by default is through time and the scale by default is the minimum to maximum of the entire range of the result.

To view the Vent region pressure result, either select the check box next to the result in the **Study Tasks** pane, or right-click the result name and select **Show**, then use the Animation tools to view the result over time.

You can control how this result is displayed by editing the plot properties.

Using this result

The Vent region pressure result shows the air pressure in areas of the part that are not filled with polymer. You can use this result together with the Air traps, including air vents result to identify regions of entrapped air that may be suitable vent locations.

Things to look for

At the end of the analysis, the vent region pressure should be zero across the entire part. This means that there should be no regions of entrapped air where pressure can build up. To optimize the placement of air vents, try one of these options.

- Move the injection location(s) so that the air traps form in easy-to-vent areas.
- Move the venting analysis location to more closely match the location of the trapped air to be vented.

Viscosity result

61

The Viscosity result shows the viscosity of the polymer at the time the results file was written.

The viscosity of a material is described as a measure of its ability to flow under an applied pressure.

This result is generated from a Fill analysis using 3D analysis technology.

Using this result

The viscosity of polymers is dependant on temperature and shear rate. In general, as the temperature and shear rate that the polymer experiences both increase, the viscosity will decrease, indicating a greater ability to flow under a given applied pressure. The results are calculated from the viscosity distribution across the molten (above transition) part of the cavity.

For a Fill analysis using 3D analysis technology, viscosity is calculated for each tetrahedral element or node throughout the model. The values calculated by the program are the actual viscosity values for the tetrahedral elements, or nodes and represent a real case injection molding scenario. The 3D viscosity result is more accurate than the fast algorithm or the finite difference algorithm because actual values are being determined at each node of the solid model.

NOTE: Use the cutting plane tool to view Viscosity results through the solid model.

Things to look for

- Areas of high viscosity, which could indicate filling problems.
- Temperature (3D) result.
- Shear rate (3D) result.

Average volumetric shrinkage result 62

The average volumetric shrinkage result shows the average value of volumetric shrinkage over the half-gap thickness for 3D models.

Volumetric shrinkage is the percentage increase in local density from the end of the packing phase to when the part has cooled to the ambient reference temperature (the default value is 25°C/77°F).

NOTE: The packing phase includes both packing time and cooling (holding) time.

Volumetric shrinkage calculations begin once the cavity is filled, based on the difference between the current pvT state and the reference state (where the pressure *p* is zero and temperature *T* is the specified ambient temperature):
volumetric shrinkage (t)=average density⊠(t)density⊠(Tambient, patm)
As the mass of an element changes (for example, with polymer flow during packing), shrinkage continues to change with each change in the element's pvT state. Once the mass stops changing, the element's current pvT state is fixed in the shrinkage calculation as the reference state.

The mass of an element stops changing when the cavity pressure has decayed to zero. After this, the volumetric shrinkage becomes a constant. However, if the holding pressure is removed before the material is frozen or while the pressure in the cavity is still non-zero, the volumetric shrinkage may rebound due to possible backflow into the nozzle or other warmer areas of the part.

Using this result

For unambiguous interpretation of the Volumetric shrinkage result, deselect the **Nodal averaged** display option. Right-click the result name and select **Properties**, then select the the **Optional Settings** tab, then deselect **Nodal averaged**.

The average volumetric shrinkage for 3D is the average value of volumetric shrinkage over the half-gap thickness, and is plotted on the surface. This result can be used to detect sink marks on your model. High shrinkage values could indicate sink marks or voids inside the part.

Volumetric shrinkage should be uniform across the whole part to reduce warpage.

NOTE: For thermoplastic materials, volumetric shrinkage should be less than the observed maximum value for the material. You can find material shrinkage information in the Thermoplastic material database. Right-click the material in the Study Tasks pane and select **Details**. The **Thermoplastics material** dialog is displayed.

Select the **Shrinkage Properties** tab. The observed maximum shrinkage values are in the **Observed shrinkage** section.

Material shrinkage information is not provided in the Thermoset material database.

Volumetric shrinkage can be controlled by the use of packing profiles.

Things to look for

- Localized areas of high shrinkage can result in internal voids or sink marks when the part cools.
- Shrinkage values should be uniform throughout the part. This is important for good packing of the material, ensuring good structural and visual integrity of the part. Use a packing profile to make the shrinkage more uniform.
- Negative volumetric values indicate expansion rather than shrinkage. Avoid negative shrinkage on ribs as this can cause ejection problems.
- Are the values in the expected range for the material?
    - Materials that shrink isotropically have a linear shrinkage which is approximately one third of the volumetric shrinkage.

      For molded materials, the linear shrinkages in the thickness, flow and transverse directions depend on the effects of relaxation and orientation.
    - For shell-like geometries, it is expected that the shrinkage in the thickness direction should be higher than the shrinkage in the plane of the part. Shrinkage in the thickness direction is likely to be greater than one third of the volumetric shrinkage, while in-plane shrinkage should be less than one third of the volumetric shrinkage. Many mold features act as constraints to in-plane shrinkage. If you are using a fiber-filled material, the orientation of the fibers in the plane of the part will limit shrinkage in this direction. Shrinkage in the thickness direction is relatively unconstrained.

Volumetric shrinkage result 63

The Volumetric shrinkage result shows the volumetric shrinkage for each node, as a percentage of the original volume.

Volumetric shrinkage is the percentage increase in local density from the end of the packing phase to when the part has cooled to the ambient reference temperature (the default value is 25°C/77°F).

NOTE: The packing phase includes both packing time and cooling (holding) time.

This result is a Pack analysis result for 3D analysis technology.

Volumetric shrinkage calculations begin once the cavity is filled, based on the difference between the current pvT state and the reference state (where the pressure *p* is zero and temperature *T* is the specified ambient temperature):
volumetric shrinkage (t)=average density(t)density(Tambient, patm)
As the mass of an element changes (for example, with polymer flow during packing), shrinkage continues to change with each change in the element's pvT state. Once the mass stops changing, the element's current pvT state is fixed in the shrinkage calculation as the reference state.

The mass of an element stops changing when the cavity pressure has decayed to zero. After this, the volumetric shrinkage becomes a constant. However, if the holding pressure is removed before the material is frozen or while the pressure in the cavity is still non-zero, the volumetric shrinkage may rebound due to possible backflow into the nozzle or other warmer areas of the part.

Using this result

This result can be used to detect sink marks on your model. High shrinkage values could indicate sink marks or voids inside the part.

Volumetric shrinkage should be uniform across the whole part to reduce warpage.

NOTE: For thermoplastic materials, volumetric shrinkage should be less than the observed maximum value for the material. You can find material shrinkage information in the Thermoplastic material database. Right-click the material in the Study Tasks pane and select **Details**. The **Thermoplastics material** dialog is displayed. Select the **Shrinkage Properties** tab. The observed maximum shrinkage values are in the **Observed shrinkage** section.

Material shrinkage information is not provided in the Thermoset material database.

Volumetric shrinkage can be controlled by the use of packing profiles.

NOTE: Use the Cutting Plane tool to view 3D Volumetric shrinkage results. A cutting plane shows volumetric shrinkage values through the solid model. This will tell you whether your solid part is likely to shrink on the inside.

Things to look for

- Localized areas of high shrinkage can result in internal voids or sink marks when the part cools.
- Shrinkage values should be uniform throughout the part. This is important for good packing of the material, ensuring good structural and visual integrity of the part. Use a packing profile to make the shrinkage more uniform.
- Negative volumetric values indicate expansion rather than shrinkage. Avoid negative shrinkage on ribs as this can cause ejection problems.
- Are the values in the expected range for the material?
  - For isotropic 3D solid materials, linear shrinkage is approximately one third of the volumetric shrinkage, where volumetric shrinkage is evenly distributed in all directions. This value can be considered an upper bound.
  - For transversely isotropic materials in thick parts, volumetric shrinkage is approximately equal to the shrinkage in the flow direction plus two times the shrinkage in the transverse direction.
  - For shell-like geometries, it is expected that the shrinkage in the thickness direction should be higher than the shrinkage in the plane of the part. Shrinkage in the thickness direction is likely to be greater than one third of the volumetric shrinkage, while in-plane shrinkage should be less than one third of the volumetric shrinkage. Many mold features act as constraints to in-plane shrinkage. If you are using a fiber-filled material, the orientation of the fibers in the plane of the part will limit shrinkage in this direction. Shrinkage in the thickness direction is relatively unconstrained.

64 Volumetric shrinkage at ejection result

The Volumetric shrinkage at ejection result shows the volumetric shrinkage for each area expressed as a percent of the original modeled volume.

Volumetric shrinkage at ejection is the decrease in local volume from the end of the cooling stage to when the part has cooled to the ambient reference temperature (the default value is 25°C/77°F).

NOTE: The packing phase includes both packing time and cooling (holding) time.

Volumetric shrinkage calculations begin once the cavity is filled, based on the difference between the current pvT state and the reference state (where the pressure *p* is zero and temperature *T* is the specified ambient temperature):
volumetric shrinkage (t)=average density(t)density(Tambient, patm)
As the mass of an element changes (for example, with polymer flow during packing), shrinkage continues to change with each change in the element's pvT state. Once the mass stops changing, the element's current pvT state is fixed in the shrinkage calculation as the reference state.

The mass of an element stops changing when the cavity pressure has decayed to zero. After this, the volumetric shrinkage becomes a constant. However, if the holding pressure is removed before the material is frozen or while the pressure in the cavity is still non-zero, the volumetric shrinkage may rebound due to possible backflow into the nozzle or other warmer areas of the part.

Using this result

For unambiguous interpretation of the Volumetric shrinkage at ejection result, deselect the **Nodal averaged** display option. Right-click the result name and select **Properties**, select the **Optional Settings** tab, and then deselect **Nodal averaged**.

This result can be used to detect sink marks on your model. High shrinkage values could indicate sink marks or voids inside the part.

Volumetric shrinkage should be uniform across the whole part to reduce warpage.

NOTE: For thermoplastic materials, volumetric shrinkage should be less than the observed maximum value for the material. You can find material shrinkage information in the Thermoplastic material database. Right-click the material in the **Study Tasks** pane and select **Details**. The **Thermoplastics material** dialog is displayed. Select the **Shrinkage Properties** tab. The observed maximum shrinkage values are in the **Observed shrinkage** section.

Material shrinkage information is not provided in the Thermoset material database.

Volumetric shrinkage can be controlled by the use of packing profiles.

Things to look for

When viewing the Best gate location result, watch for the following:

- Localized areas of high shrinkage can result in internal voids or sink marks when the part cools.
- Shrinkage values should be uniform throughout the part. This is important for good packing of the material, ensuring good structural and visual integrity of the part. Use a packing profile to make the shrinkage more uniform.
- Negative volumetric values indicate expansion rather than shrinkage. Avoid negative shrinkage on ribs because this can cause ejection problems.
- Check that values are in the expected range for the material, noting the following:
  - Materials that shrink isotropically have a linear shrinkage that is approximately one third of the volumetric shrinkage.

    For molded materials, the linear shrinkages in the thickness, flow, and transverse directions depend on the effects of relaxation and orientation.
  - For shell-like geometries, the shrinkage in the thickness direction should be higher than the shrinkage in the plane of the part. Shrinkage in the thickness direction is likely to be greater than one third of the volumetric shrinkage, and in-plane shrinkage should be less than one third of the volumetric shrinkage. Many mold features act as constraints on in-plane shrinkage. When fiber-filled material are used, the orientation of the fibers in the plane of the part will limit shrinkage in this direction. Shrinkage in the thickness direction is relatively unconstrained.

Weld lines result

65

The Weld lines result displays the angle of convergence as two flow fronts meet. The presence of weld lines may indicate a structural weakness and/or a surface blemish.

TIP: By overlaying the Fill time result and stepping through the animation, you can see how the flow fronts converge.

The term "weld line" is often used to mean both weld and meld lines. The only difference between them is the angle at which they are formed; weld lines form at lower angles than meld lines. Weld lines can cause structural problems and make the part visually unacceptable, but they are unavoidable when the flow front splits and comes together around a hole, or if the part has multiple gates.

Consider the processing conditions and position of the weld lines to determine whether the weld lines will be high quality. Weld and meld lines should be avoided, particularly weld lines in areas that require strength or a smooth appearance.

NOTE: For Midplane or Dual Domain studies, this result is the same as a custom weld line plot with a meeting angle of 135 degrees. For a different meeting angle, create a custom weld line plot and specify the meeting angle.

Processing conditions help to determine the quality of weld or meld lines. Weld line strength is influenced by the temperature at which the weld line is formed and the pressure exerted on the weld until the part freezes; pressure is 0 at the weld line. Typically a "good" weld will occur if the temperature of the melt at the weld line as it forms is no more than 20°C below the injection temperature.

Using this result

To examine the processing conditions under which the weld lines occurred, you can change the weld line properties.

Weld lines can be moved by changing the fill pattern to make the flow fronts meet at a different place. To move weld lines:

- Alter the gate locations.
- Change the thickness of the part.

To improve the quality of weld lines:

- Increase the melt temperature, injection speed, or packing pressure. This will enable the flow fronts to weld to each other more effectively.
- Increase the diameters of gates and runners to make it easier to pack the part.

- Move injection locations to make weld lines form closer to the gates. The weld line is then created with a higher flow front temperature and is packed with more pressure.
- Move injection locations to make flow fronts meet more obliquely, turning the weld line into a meld line.
- Place a vent in the area of the weld line. This will remove air traps, which could further weaken the weld line.
- Optimize the design of the runner system.
- Reduce runner dimensions and maintain the same flow rate. Shear heating can then be utilized to increase the melt temperature at the flow front.

Solving one problem can introduce other problems to the injection molding process. Carefully consider all the relevant aspects of the mold design specification before choosing an option.

NOTE: Accurate weld line prediction depends on the quality of the mesh. Refining the mesh can improve weld line prediction, especially around holes.

Things to look for

The Weld Lines result helps you identify the following problems:

- Structural problems: The part may be more likely to fracture or deform at a weld line, especially if the weld line is of a low quality. This weakness is a more serious problem in areas of the part that are subject to stress.
- Visual defects: A weld line can cause a line, notch, or color change on the surface of the part. If the weld line is positioned on a non-critical part surface, such as the bottom, this may not be a problem.

Weld lines result

You can change the Plot properties of the display to examine the processing conditions under which the weld lines occurred.

Customizing weld line display properties

Weld lines are created when two or more flow paths meet during the filling process. Weld line strength is influenced by the temperature at which the weld line is formed (not more than 20°C below injection temperature) and the pressure exerted on the weld while the material is within its recommended processing temperature range (the higher the pressure, the more the material is rediffused).

1. Select **Weld lines** from the **Results** section of the **Study Tasks** pane.
2. Click **Results tab > Properties panel > Plot Properties**, and select the **Highlight** tab from the top of the dialog box that appears.
3. From the **Color** box, click **Browse**.

4 From the dialog that appears, select the required feature and then click **OK** twice.

5 The scale bar in the **Model Pane** now displays the extent of results for the feature selected.

6 Individual results can be obtained by selecting **Results tab > Plots panel > Examine** and then clicking on the part of the weld line of interest.

These results should then be compared to the processing parameters used for this analysis.

Creating a custom weld line angle result

Specifying a custom weld line angle allows you to see the effect of the meeting angle on the length and severity of the weld line.

The **Weld Lines** custom plot is available for Midplane and Dual Domain studies. It allows you to specify the angle at which the flow fronts meet and create a weld line.

1 Click (**Results tab > Plots panel > New Plot > Custom**).

2 Select **Weld Lines** from the **Create Custom Plot** pane.

3 Enter a name for the plot in the **Plot Name** text box.

4 Enter a meeting angle into the **Max. meeting angle** text box.

The default meeting angle is 135 degrees.

5 Click **OK**.

The plot is displayed and the plot name is listed in the **Study Tasks** pane under **User-Defined Plots**.

You can create several plots with different meeting angles to compare the effects. Click the check box next to each plot name to display each plot in turn and compare the effects of the meeting angle.

TIP: To assist in comparing plots, make the color of the weld lines different in each plot. You may also find it useful to overlay plots to compare the weld lines.

Weld and meld lines result

66

The Weld and meld lines result shows where weld and meld lines are likely to occur on your model.

This result is generated at the end of a Fill analysis.

Weld and meld lines are a weakness or visible flaw created when two or more flows meet and converge while filling a part. The presence of weld or meld lines may indicate a structural weakness and/or a surface blemish. The only difference between a meld line and weld line definition is the angle at which they are formed.

A meld line is typically formed by parallel flows. Meld lines tend to be stronger and generally less visible than weld lines. The quality of the meld line is dependent on the material type, the type and amount of fillers, and the pressure and temperature at the meld line.

Weld lines are formed at lower angles. Weld lines can cause structural problems, and they can also make the part visually unacceptable. Therefore weld and meld lines should be avoided if possible. However, weld lines are unavoidable when the flow front splits and comes together, around a hole, or has multiple gates. Look at the processing conditions and the weld line position to decide if the weld lines will be of a high quality. Avoid weld lines in areas which need strength, or need to appear smooth.

NOTE: This result is the same as a custom weld line plot with a meeting angle of 179 degrees. For a different meeting angle, create a custom weld line plot and specify the meeting angle.

Processing conditions help to determine the quality of weld or meld lines. Weld line strength is influenced by the temperature at which the weld line is formed and the pressure exerted on the weld until the part freezes, (pressure is 0 at the weld line). A good weld typically occurs when the melt temperature where the weld line is formed is no more than 20°C below the injection temperature.

Using this result

Weld and meld lines can cause structural problems, and they can also make the part visually unacceptable. However, some weld and lines are unavoidable, so you need to look at the processing conditions and the weld/meld line position to decide if they will be of a high quality.

To examine the processing conditions under which the weld or meld lines occurred, you can change the weld line properties.

To move weld lines, change the fill pattern to make the flow fronts meet at a different place:

- Alter gate positions.
- Change the part thickness.

To improve the quality of weld lines:

- Increase the melt temperature, injection speed, or packing pressure. This will allow the flow fronts to weld to each other better.
- Increase the diameters of gates and runners, to make it easier to pack the part.
- Move injection locations to make weld lines form closer to the gates, so the weld line is created with a higher flow front temperature and is packed with more pressure.
- Move injection locations to make flow front meet more obliquely, turning the weld line into a meld line.
- Place a vent in the area of the weld line. This will remove air traps, which could further weaken the weld line.
- Optimize runner system design.
- Reduce runner dimensions and maintain the same flow rate to use shear heating to increase the melt temperature at the flow front.

Solving one problem can often introduce other problems to the injection molding process. Each option requires consideration of all relevant aspects of the mold design specification.

NOTE: Accurate weld or meld line prediction depends on the quality of the mesh. Refining the mesh can improve weld line prediction, especially around holes.

Things to look for

This result helps you identify the following problems:

Structural problems	The part may be more likely to fracture or deform at a weld line, especially if the weld line is of a low quality. This weakness will be more of a problem in areas of the part which will be subject to stress.
Visual defects	A weld line can cause a line, notch or color change on the surface of the part. If the weld line is positioned on a non-critical part surface (for example, the bottom of the part), this may not be a problem.