Autodesk® Moldflow® Communicator 2012

Troubleshooting guide

Revision 1, 1 April 2011.

Contents

Troubleshooting guide

1

Molding problems fall into three main categories; modeling, machine settings, and materials. The effects of these choices is far-reaching, ranging from visual defects such as sink marks, so physical defects such as warpage and even financial implications resulting from excessive cycle times or waste from short shots.

Cooling problems

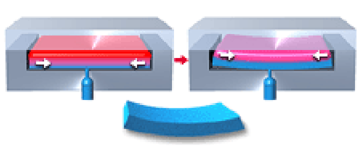
Cooling accounts for 80 percent of the molding cycle and is a crucial phase in successful molding. Critical dimensions, warpage, and surface finish are all affected by cooling conditions, and poorly cooled parts increase cycle time, dimensional problems, and wastage.

The following sections list common cooling problems and their solutions.

Non-uniform cooling

The main problem caused by non-uniform cooling is warpage.

The following diagram shows how the bottom layer cools first. As this layer shrinks it slides under the top layer which is still molten. As the top layer freezes it becomes fixed to the shrinking bottom layer and it cannot slide over it. The resulting tensile stress can cause the part to warp.



What to do

The acceptable temperature difference between the top and bottom sides of an area depends on how easily the part will warp. For example, a part with ribs will be less likely to warp. The geometry of the part is one factor that determines the acceptable temperature difference between the top and bottom of an area.

Consider the following specific areas in the mold:

- Hot spots—try to achieve more uniform cooling
- Spacing between the cooling channel and mold cavity.
- Spacing between adjacent cooling channels.

Excessive cycle time

The cycle time is the time taken to produce a molded part, which is measured from the start of injection until the end of clamp open time. High cycle times reduce productivity and increase costs.

There will always be a compromise between uniform cooling to assure part quality and fast cooling to minimize part costs. The function of the part will determine whether speed or quality is more important.

Excessive cycle times can be caused by the following conditions:

- High mold and melt temperatures
- Poor cooling system design
- Excessive filling and packing times
- Fast injection rates causing long cooling times

What to do

To reduce excessive cycle time:

- Optimize the filling time
- Optimize the packing time
- Use minimal cooling time
- Optimize the cooling system
- Change the part geometry to cool faster and more uniformly

All of these remedies should be evaluated in conjunction with each other, and with the quality and cost requirements of the part. Compromises may be necessary.

Insufficient pump capacity

The heat transfer rate is reduced when the flow is laminar instead of turbulent because of an insufficient pump capacity.

What to do

The available pump capacity must be greater than that required by the cooling circuits in the mold. The amount of heat that has to be removed can be calculated from the weight of the part and its enthalpy at molding temperature, allowing for the fact that the temperature of the molding when ejected will be generally well above ambient.

Flow problems

Flow problems manifest themselves in a variety of ways, including overpacking, insufficient packing, and visual defects.

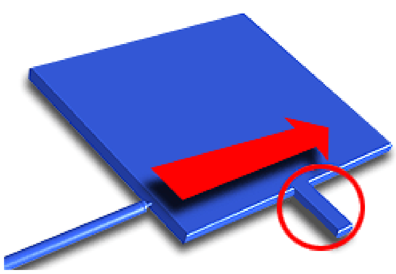
Hesitation

Hesitation, which occurs when the melt flow slows down or stops along a particular flow path, can lead to asymmetrical and unpredictable flow patterns.

Hesitation

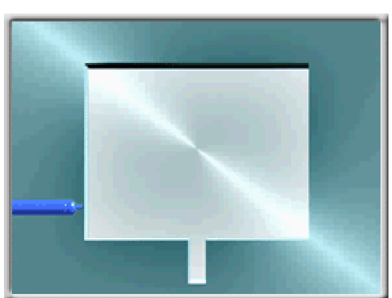
When the melt entering a cavity is filling a thin section and a thick section, it tends to fill the thick section first because this route is less resistant to flow. This can result in the melt in the thin section stopping or slowing significantly. Hesitation can reduce part quality due to variations in surface appearance, poor packing, high stresses and non-uniform orientation of the plastic molecules. If the hesitation enables the flow front to freeze completely, part of the cavity may remain unfilled, resulting in a short shot.

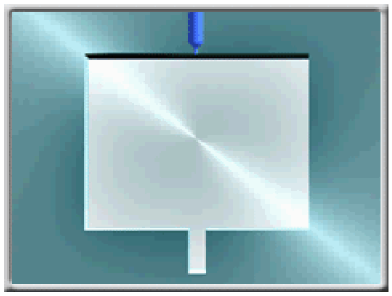
The melt cools more rapidly as it starts to slow down so the viscosity increases. In turn, this higher viscosity inhibits flow further, causing even faster cooling, so the problem is self-propagating. Hesitation can occur in ribs and in thin sections of parts that have significant changes in wall thickness. In the following diagram, the rib circled in red offers a higher resistance to flow because it is much thinner than the rest of the part.



What to do

The following animations show how hesitation in a part can be avoided by using a different injection location. When there is no alternative route available, the flow in the rib will be continuous and not hesitate, as shown in the second example.





Viewing the fill time and temperature results can help explain why the hesitation occurred. The fill time plot will show hesitation by a narrow spacing of fill time contours, and the temperature plot will show hesitation by a low temperature and a large temperature gradient.

Hesitation can also be reduced by taking the following steps:

- Move the polymer injection location away from the area of hesitation so that the bulk of the cavity fills before the melt reaches the thin area. The absence of alternative flow paths will give less time for the polymer to hesitate.
- Move the polymer injection location to a place that will cause greater pressure to be applied where the hesitation occurred. It is useful to have thin ribs/bosses as the last point to fill, so all the injection pressure is applied at this point.
- Increase the wall thickness where the hesitation occurred, to reduce the resistance to flow.
- Use a less viscous material (that is, a material with a higher melt flow index).
- Inject more quickly to reduce the potential of hesitation time.
- Increase the melt temp so that it flows into the thin area more readily.

NOTE: Changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Volumetric shrinkage

Volumetric shrinkage is the contraction of polymer due to the change in temperature from melt temperature to ambient temperature.

High volumetric shrinkage can cause part warpage, sink marks, critical dimensions that are too small, and internal voids. Excessive wall thickness and inadequate packing can both contribute to high volumetric shrinkage in a part.

What to do

The key result to use to identify high volumetric shrinkage are the **shrinkage** results.

To reduce volumetric shrinkage, you can alter:

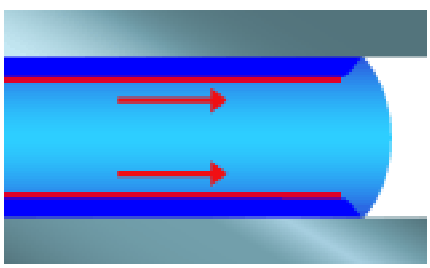
- Part design, such as the wall thickness.
- Mold design, such as gate positions.
- Processing conditions, by increasing packing pressure.

NOTE: Alterations such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Shear stress

Shear stress is stress applied to the melt as it flows through the mold. This stress is caused by excessive friction between layers of plastic flowing in the cavity.

This type of stress can occur when the material viscosity is high and/or the material flow rate is high. Shear stress often manifests around the gate due to high injection speeds, and at the end of flow if a constant flow rate is used. High shear stress can cause the plastic to degrade and fail due to stress cracks. The red line in the following diagram represents the area of maximum shear stress. This is between the dark blue frozen layer and the light blue melt.



What to do

To reduce internal shear stress:

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

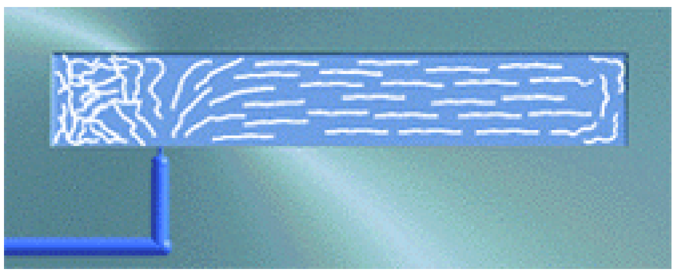
Overpacking

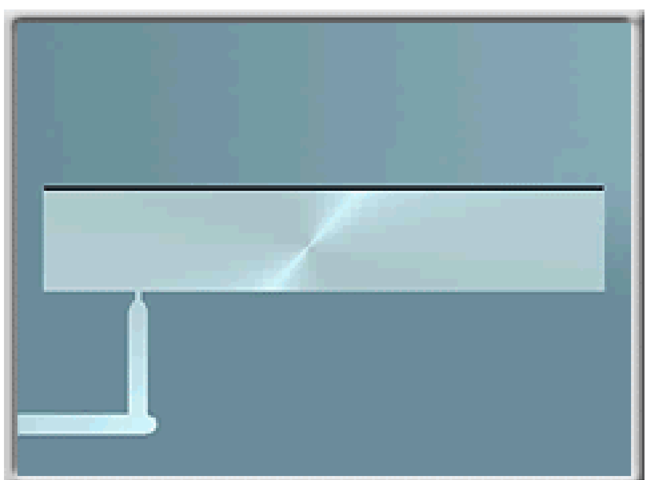
Overpacking occurs when extra material is compressed in one flow path while other flow paths are still filling.

Overpacking

Overpacking, which usually occurs in sections with the shortest fill time, can cause a range of problems, including warpage due to non-uniform shrinkage, increased part weight due to wasted material, and non-uniform density distribution throughout the part.

Overpacking occurs when the easiest, that is the shortest or thickest, flow paths fill first. When these flow paths have filled, they are still under pressure as extra plastic is injected into the cavity to fill the remaining flow paths. This pressure pushes more material into the already full flow path, causing it to have a higher density and lower shrinkage than other regions. The overpacked fill path freezes under pressure so stresses are frozen in. In the following diagram, the white lines represent the polymer molecules. Note that the flow paths are not balanced and overpacking will occur to the left side of the part.





What to do

The key result that identifies overpacking is the fill time result. Display the fill time at 100 percent fill, and look for any flow paths that do not finish at the same time as the first path. Solving overpacking requires a rebalancing of the flow paths, which can be achieved by the following:

- Thicken or make thinner the parts of the model that act as flow leaders or deflectors.

- Move the injection location to a position that will define similar length flow paths.
- Divide the cavity into imaginary sections, and use one injection location for each section.
- Remove unnecessary gates.

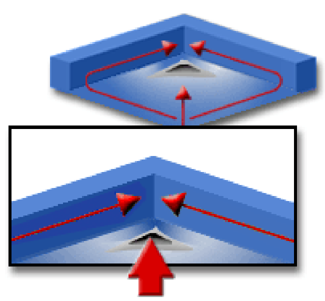
Racetrack effect

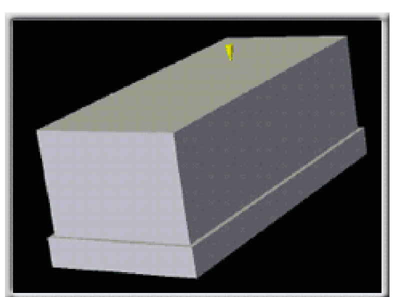
The racetrack effect occurs when flow races through thick sections of the cavity before the thin sections have filled.

NOTE: Thick sections offer less resistance to flow than thin sections.

Racetrack Effect

The racetrack effect indicates unbalanced flow paths and can often cause unnecessary weld lines and air traps. The following diagram shows a part with a thick rim.

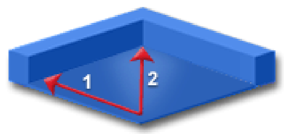




The flow of plastic (red arrows) races around the rim, trapping a pocket of air (blue circle).

What to do

A large difference in wall thickness throughout a part can cause problems, but is sometimes necessary from a design point of view. However, in the previous example, the racetrack effect through the thick regions is not actually the problem. The problem is unbalanced flow that allows the racetrack effect to occur. If the plastic reached all parts of the thick rim at the same time, the racetrack effect would not occur.



Flow path 1 is shorter than flow path 2. However, by slightly thickening flow path 2 or thinning flow path 1 (see *How thickness affects flow*), the plastic could be forced to reach all parts of the thick rim at the same time. This would result in balanced flows.

Unbalanced flow

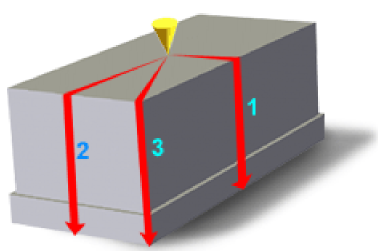
Unbalanced flow is plastic completely filling some flow paths in the mold before other flow paths have filled.

Unbalanced flow can be the cause of many molding problems such as flashing, short shots, high cycle time, density differences throughout the part, warpage, air traps and extra weld lines.

Flow is balanced when all the extremities of the mold fill at the same time.

To recognize unbalanced flow, you need to recognize the different flow paths in the mold. These are the different routes that the plastic takes throughout the cavity.

The following part contains three fundamental flow paths (shown as red arrows).



Each flow path is of a different length. Therefore, if the part has uniform thickness, flow path 1 will fill first, followed by flow path 2, followed by flow path 3.

To identify unbalanced flow, use either the fill preview or the fill time result.

What to do

By altering the thickness of regions within the part, flow can be hastened or delayed in certain directions to help balance flows. In the above diagram, varying the part thickness and creating flow leaders and deflectors, thinning flow path 1 and thickening flow path 3, is the answer. These variations in thickness are known as Flow Leaders or Flow Deflectors.

In other examples it is often necessary to consider the position of the polymer injection location, or the number of polymer injection locations.

For example, if you choose a single injection location that defines some flow paths to be three or four times the length of others, then it is almost impossible to balance flows. Try moving the polymer injection location to a position that will define similar length flow paths. Alternatively, visualize the cavity in smaller, more manageable sections. Then use multiple injection locations, one per sub-section.

For a multi-cavity part, balance flows in each cavity first, then proceed to alter runner dimensions to ensure that:

- All cavities fill at approximately the same time, with the same pressure.
- The temperature at the end of fill shows uniform distribution in each cavity, predicting uniform shrinkage, and acceptable weld line quality.
- The shear stress in each cavity (ignore runners) is less than the recommended limit for the material chosen.

Underflow

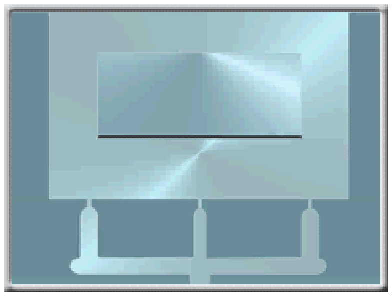
Underflow is when a flow front reverses direction.

It occurs when flow fronts from two directions meet, pause momentarily, then one of the flows reverses direction and flows back between the outer frozen layers. When the flow reverses direction the frozen layer partly re-melts due to shear heating.

In the animation below, the gate in the center has a much smaller volume to fill than the other two gates. When the center gate's volume is filled, the other volumes are still filling, so the flow front from the left gate has a lower pressure than the center gate. When the two flow fronts meet, the left flow front reverses direction.

The arrow shows the direction of the underflow.

This flow reversal gives poor part quality, both from surface appearance and structural viewpoints.



What to do

Inspect the filling pattern to assess if underflow is likely to occur. Displaying the fill time result at 100% will not indicate the presence of underflow. Play the fill time animation from beginning to end and watch for any flow fronts meeting, then consider the geometry surrounding this point.

Molding problems

Molding problems manifest themselves in many ways, from visual defects like burn marks, to physical problems such as delamination. This category of defects covers that broad group not caused by primarily by cooling or flow problems.

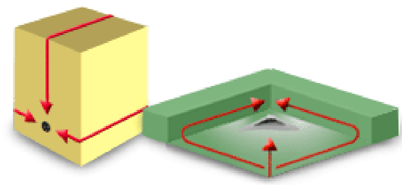
Troubleshooting air trap problems

Air traps occur when converging flow fronts surround and trap a bubble of air.

The trapped air can cause incomplete filling and packing, and will often cause a surface blemish in the final part. Air trapped in pockets may compress, heat up and cause burn marks.

Causes

- Racetrack effect. The racetrack effect occurs when molten plastic flows into thicker regions more easily than thin regions. The flow divides and then fills thicker sections before combining again to fill the thinner sections. The recombined flow can reverse to meet the oncoming flow in the thinner section.
- Hesitation. In a part with multiple flow paths, the flow can slow down or hesitate in thin regions.
- Unbalanced flow paths. Flow paths do not need to exhibit the racetrack effect or hesitation to have unbalanced flow. In a part with uniform thickness, the physical length of flow paths may vary, and again, air traps may occur.
- Inadequate venting. Lack of vents or undersized vents in these last-to-fill areas are a common cause of air traps.



Remedies

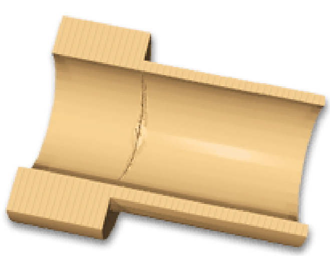
- Balance flow paths.
- Avoid hesitation and racetrack effects.
- Balance runners. Changing the runner system can alter the filling pattern in such a way that the last-to-fill areas are located at the proper venting locations.
- Vent appropriately. If air traps do exist, they should be positioned in regions that can be easily vented or ejection and/or vent pins added so that air can be removed.

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

Troubleshooting brittleness problems

A brittle molded part has a tendency to break or crack.

Brittleness results from shorter molecular chain length (thus lower molecular weight). As a result, the physical integrity of the part is substantially less than the specification.



Causes

- Material degradation. This can be caused by excessive injection speed, residence time or melt temperature. Improper screw or runner system design may also lead to material degradation.
- Weld line weaknesses.
- Non-optimal crystallinity.

- High residual stress.
- Incompatible materials blended together.
- Too much regrind.
- Improper drying conditions. Excessive drying either drives off volatiles in the plastic, making it more sensitive to processing, or degrades the material by reducing the molecular weight.

Remedies

- Set proper drying conditions before molding. Material suppliers can provide optimum drying conditions for the specific materials.
- Reduce the regrind material. Contact material suppliers to get the recommended levels of regrind to use.
- Use a different material.
- Optimize the runner system design. Restrictive sprue, runner, gate, or even part design could cause excessive shear heating that aggravates an already overheated material, causing material degradation.
- Modify the screw design. Contact material/machine suppliers to get the right screw design information to avoid improper melt mix or overheating that leads to material degradation.
- Select a machine with smaller shot size. Minimizing residence time reduces material degradation.
- Reduce the residual stress.
- Strengthen the weld lines. Increase melt temperature within limits, not to overheat the material.

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

NOTE: Changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

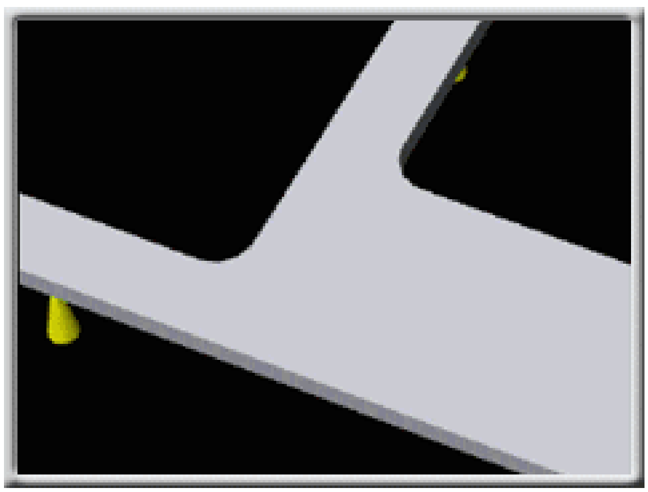
Troubleshooting burn marks problems

Burn marks are small, dark or black spots on the part's surface.

This phenomena is also often referred to as dark streaks or specks.

Causes

- Adiabatically heated trapped air. Air trapped in pockets may compress, heat up and cause burn marks.



- Material degradation. This can be caused by excessive injection speed, residence time or melt temperature. Improper screw or runner system design may also lead to material degradation.

Remedies

- Eliminate air traps. To prevent burn marks, move air traps to places which can be vented, or where ejector pins can be added.
- Optimize the runner system design. Restrictive sprue, runner, gate, or even part design could cause excessive shear heating that aggravates an already overheated material, causing material degradation.
- Modify screw design. Contact material/machine suppliers to get the right screw design information to avoid improper melt mix or overheating that leads to material degradation.
- Select machine with smaller shot size.
- Optimize melt temperature. Reduce temperature to prevent material degradation from overheating. Note that residual stress may increase when the melt temperature is reduced.

- Optimize back pressure, screw rotation speed, or injection speed. Balance shear heat against residual stress.

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

Troubleshooting cracking problems

Cracking can cause part failure, a short part life and be visually unacceptable.

Cracks may not be obvious until several days or weeks after production. Hence, it is better to recognize and remove the potential problem of cracking before production. Autodesk Moldflow Insight provides detailed shear stress results.

Causes

- High residual stresses. Cracks may occur in regions where internal shear stresses are frozen into the part.
- Weld line weaknesses. When two or more flow paths meet during a filling process, structural problems and/or visibly unacceptable results will occur.
- Differential shrinkage. Differential orientation, packing and cooling cause differential shrinkage resulting in high internal stress levels being frozen in.

Remedies

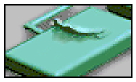
- Minimize residual stress. Program the ram speed or increase wall thickness to reduce flow induced stresses. Check for the recommended maximum shear stress value for the material (recorded in the Autodesk Moldflow materials database)
- Minimize differential shrinkage.

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

NOTE: Changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting delamination problems

Delamination, sometimes referred to as lamination or layering, is a defect in which the surface of a molded part can be peeled off layer by layer.



Causes

- High shear stress.
- Incompatible materials blended together.
- Excessive use of mold release agent.
- Excessive material moisture. Excessive moisture heats up and forms steam, which results in delamination on the surface.
- Material degradation. This can be caused by excessive injection speed, residence time or melt temperature. Improper screw or runner system design may also lead to material degradation.

Remedies

Eliminate Degradation and Excessive Shear Stress

- Reduce shear stress.
- Remove excessive moisture. Material suppliers can provide optimum drying conditions for the specific materials.
- Reduce regrind material.
- Avoid excessive use of mold release agent. Repair the ejection system or other problems to eliminate the difficulty of de-molding instead of over-using the mold release agent.
- Avoid material contamination.

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

NOTE: Changes to design parameters such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting dimensional variation problems

Dimensional variation is characterized by the molded part dimension varying from batch to batch, or from shot to shot while the machine settings remain the same.

Causes

- Inconsistent shrinkage. Resulting from:

1. Material variations such as property variations, varying moisture content, inconsistent melt and pigmentation;
2. Process conditions variations such as inconsistent packing and varying mold and melt temperatures;
3. Machine variations such as a damaged check ring and unstable controller.

- Narrow molding window.

Remedies

- Remove excessive moisture. Material suppliers can provide optimum drying conditions for the specific materials.
- Reduce regrind material. Contact the material suppliers to get the recommended levels of regrind to use.
- Optimize the runner system design. Poor design could cause material degradation through shear heating or inconsistent packing.
- Replace the check ring if it is broken or worn out.
- Ensure uniform mold temperature. Make sure the mold temperature is uniform by checking the cooling system.
- Set processing conditions within the molding window.
- Reduce differential shrinkage.

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

NOTE: Changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting discoloration problems

Discoloration is a color defect characterized by a molded part's color having changed from the original material color.

Causes

- Material degradation. This can be caused by excessive injection speed, residence time or melt temperature. Improper screw or runner system design may also lead to material degradation.

Remedies

- Optimize the runner system design. Restrictive sprue, runner, gate, or even part design could cause excessive shear heating that aggravates an already overheated material, causing material degradation.

- Modify screw design. Contact material suppliers to get the right screw design information to avoid improper melt mix or overheating that leads to material degradation.
- Select machine with smaller shot size. The typical shot size should be between 20 and 80 percent of machine injection capacity. For temperature-sensitive materials, the range should be narrowed down, depending on the material. Autodesk Moldflow Insight can help you select the right size machine for a specific mold. This will help avoid material remaining in the heated barrel for prolonged periods of time.
- Optimize melt temperature. Reduce temperature to avoid material degradation from overheating, or increase it to limit residual stress.
- Optimize back pressure, screw rotation speed, or injection speed. Balance shear heat against residual stress.
- Vent appropriately. Use the material supplier recommended venting size.

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

NOTE: Changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting part weight problems

In most cases, excessive part weight is an undesirable molding characteristic.

It increases production cost due to the long cycle time required for cooling the excess material and the additional cost of the material.

Causes

- Overpacking.
- Unnecessarily thick wall section.

Remedies

- Avoid overpacking.
- Use thinner wall sections with ribs. Thicken only those wall sections that require extra material for structural stability and that cannot be strengthened using another method.
- Design a part to be made by gas injection molding.

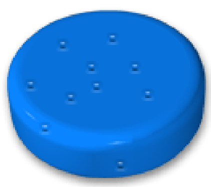
NOTE: When attempting to balance flows by altering the thickness along particular flow paths, try to use flow deflectors rather than flow leaders to keep part weight down.

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

NOTE: Changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting fish eyes problems

Fish eyes are a surface defect that results from unmelted material being pushed with the melt stream into the cavity and appearing on the surface of a molded part.



Causes

- Low melt temperature. If the melt temperature is too low to melt the material completely, the unmelted pellets will merge with the melt stream, marring the surface of the part.
- Too much regrind. The shape and size of regrind is irregular compared with original material, and can trap more air and cause the material to blend unevenly.
- Incompatible materials blended together.
- Low screw rotation speed. If the screw rotation speed and the back pressure setting are set too low, there might not be enough shear heating to melt the material completely in the barrel before the injection.

Remedies

- Reduce regrind material. Contact material suppliers to get the recommended levels of regrind to use.
- Optimize melt temperature.
- Modify screw design. Contact material suppliers to get the right screw design information to avoid improper melt mix or overheating that leads to material degradation.

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

NOTE: Design changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting flashing problems

Flashing occurs when a thin layer of material is forced out of the mold cavity at the parting line or ejector pins location.

This excess material remains attached to the molded article, and normally has to be manually removed.

Causes

- Worn or poorly fitted cavity/mold plates. Possible causes include mold plate deformations and obstructions (grease, dirt, debris).
- Insufficient clamp force. The machine clamp force must be greater than the pressure in the cavity (that is, clamp opening force), to sufficiently hold the mold plates shut.
- Overpacking. Overpacked sections cause increased localized pressure.
- Non-optimal molding conditions. Possible causes include material viscosity, injection rate, and runner system design. For example, a high melt temperature results in a less viscous melt.
- Improper venting. Examples include an improperly designed venting system, an ineffective venting system, or a venting system that is too deep.

Remedies

- Ensure the mold plates are correctly fitted, and set up the mold so that it seals properly. Clear any obstructions from the machine. If deformation of a mold plate occurs during the molding process, add a pillar support or thicken the mold plates.
- Avoid overpacking.
- Select a machine with higher clamp force capability.
- Vent appropriately. Use the material supplier's recommended venting size.
- Optimize processing conditions. Reduce pressures and shot size to the minimum required.

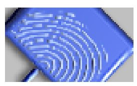
Solving one problem can often introduce other problems to the injection molding process. Each option hence requires consideration of all relevant aspects of the mold design specification.

NOTE: Design changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting flow marks problems

A flow mark or halo, is a surface defect in which circular ripples or wavelets appear near the gate.

Ripples, a similar defect, appear as small fingerprint-like waves near the edge or at the end of the flow.



Causes

- Material freezing near the gate. Low melt or mold temperature, and low ram speed can result in cold material entering the cavity. This can cause the partly solidified material to take on the form of the flow pattern.

  a normal fountain flow with no ripples, **b** flow causing ripples (R).

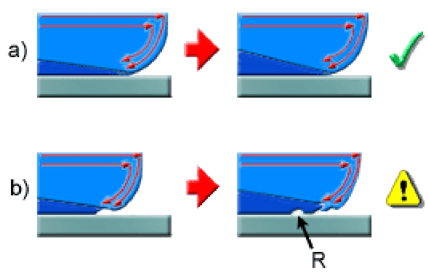
  

  Figure 1: Ripples caused by low temperature

- Insufficient material compensation. Early gate freeze-off or low packing pressure may not pack the cavity properly. The material near the gate then freezes while maintaining the form of the flow pattern.

Remedies

- Optimize the cold well. Design the cold well in the runner system to trap the cold material during the filling phase. The proper length of the cold well is usually equal to that of the runner diameter.
- Optimize the runner system design. A restrictive runner system design can result in premature gate freeze-off. It can however, increase shear heating for better melt flow.

- Increase the mold and melt temperature.
- Optimize packing pressure.

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

NOTE: Design changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting jetting problems

Jetting occurs when polymer melt is pushed at a high velocity through restrictive areas, such as the nozzle, runners, or gates; or into open, thicker areas, without forming contact with the mold wall.

The buckled, snake-like jetting stream causes contact points to form between the folds of melt in the jet, creating small scale "welds".

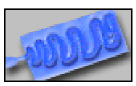


Figure 2: Jetting

Jetting leads to part weakness, surface blemishes, and a multiplicity of internal defects.

Causes

- Excessive ram speed.
- Poor gate position. Lack of melt contact with the mold allows jetting to occur.
- Inadequate hot runner system design.

Remedies

- Optimize gate design and position. Direct the melt against a metal surface by repositioning the gate or use an overlap or a submarine gate.

  Use a tab or fan gate to slow down the melt with a gradually divergent flow area. This reduces the melt shear stress and shear rate.

  a overlapping gate.

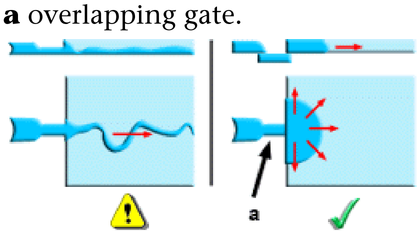
  

  Figure 3: Overlapping gate to minimise jetting

- Optimize the ram speed profile. Use an optimized ram speed profile so that melt front velocity is initially slow when the melt passes through the gate, then increases once a dispersed flow is achieved.

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

NOTE: Design changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting short shot problems

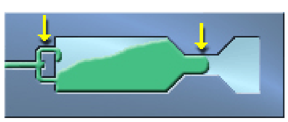
A short shot is the incomplete filling of a mold cavity which results in the production of an incomplete part.

If a part short shots, the plastic does not fill the cavity. The flow freezes off before the flow paths have completely filled.

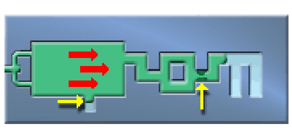
To ensure the finished part is of good quality, the part must also be adequately packed with plastic. Therefore the question to ask is not only, “"Will the part fill?”" but also, "“Can a good quality part be made?”"

Causes

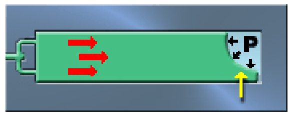
- Flow restrictions. Due to channels freezing or inadequate runner design.



- Hesitation and long or complex flow paths.



- Inadequate venting. Back pressure due to unvented air traps can cause a short shot.



- Low melt and/or mold temperatures.
- Insufficient material entering the cavity. An undersized machine, low shot volume, or inadequate ram speed.
- Machine defects. Including an empty hopper, blocked feed throat, or a worn non-return (check) valve that causes loss of pressure or volume leakage.

Remedies

Before you try one of the methods listed below, check all of the other results, so that you know the exact cause of the short shot.

- Avoid hesitation.
- Eliminate air traps. If air traps do exist, they should be positioned in areas that can be easily vented or ejection pins added so that air can be removed.
- Increase mold and melt temperature. This will decrease the viscosity of the melt, making it easier for the plastic to flow through the part.
- Increase ram speed. This can cause greater shear heating, which decreases the viscosity of the melt, making it easier for the plastic to flow through the part.
- Change the part geometry. Balance flow paths so they fill in an equal time and an equal pressure. You may need to thicken thin sections, or reduce the complexity of a flow path.
- Use a different material. Select a less viscous material (higher melt flow rate). By choosing a material with a higher melt flow rate, less injection pressure will be required to fill the part.
- Increase the maximum injection pressure for this part.

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

Troubleshooting sink marks and voids

Sink marks and voids both result from localized shrinkage of the material at thick sections without sufficient compensation.

Sink Marks

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.

Voids

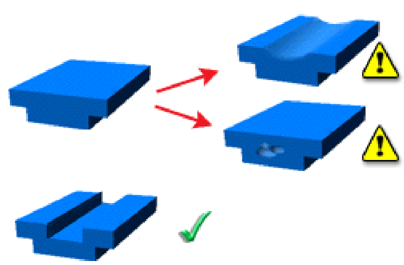
Voids are holes enclosed inside a part. These can be a single hole or a group of smaller holes. Voids may have severe impact on the structural performance of the part.

Causes

Sink marks are caused mainly by thermal contraction (shrinkage) during cooling. After the material on the outside has cooled and solidified, the core material starts to cool. Its shrinkage pulls the surface of the main wall inward, causing a sink mark. If the skin is rigid enough, deformation of the skin may be replaced by formation of a void in the core.

- Localized geometric features. sink marks typically occur in moldings with thicker sections, or at locations opposite from ribs, bosses or internal fillets.
- High volumetric shrinkage.
- Insufficient material compensation. Early gate freeze off or low packing pressure may not pack the cavity properly.
- Short packing or cooling time.
- High melt and/or mold temperatures.

Voids are caused when the outer skin of the part is stiff enough to resist the shrinkage forces thus preventing a surface depression. Instead, the material core will shrink, creating voids inside the part.



Remedies

- Optimize packing profile. As sink marks occur during packing, the most effective way to reduce or eliminate them is to control the packing pressure correctly. To determine the effects of packing on sink marks, use a simulation package such as Autodesk Moldflow Insight.
- Change the part geometry. Change the part design to minimize thick sections and reduce the thickness of any features that intersect with the main surface.

- Reduce volumetric shrinkage.
- Relocate gates to problem areas. This allows these sections to be packed before the thinner sections between the gate and the problem areas freeze.
- Optimize the runner system design. Restrictive runner system design can result in premature gate freeze off.
- Use a different material.

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

NOTE: Changes such as those above can only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

Troubleshooting weld lines and meld lines

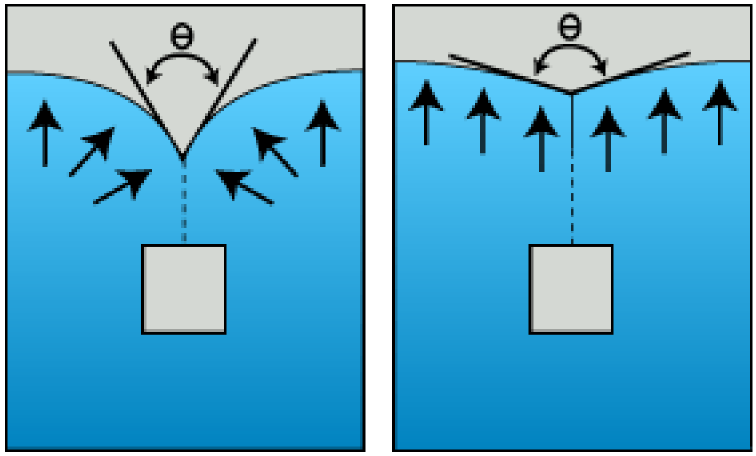
A weld or meld line on plastic parts can cause structural problems and/or be visibly unacceptable.

A weld or meld line is a weakness or visible flaw created when two or more flow paths meet during the filling process. Weld lines can be caused by material flowing around holes or inserts in the part, multiple injection gates or variable wall thickness where hesitation or "race tracking" can occur. If the different flow fronts have cooled before meeting, they don't interfuse well and can cause a weakness in the molded part. A line, notch and/or color change can appear.

NOTE: The **Weld lines** result in the **Study Tasks pane** may not show all weld lines if the model mesh is too coarse.

Difference between weld and meld lines

The difference between a weld and meld line is determined by the angle at which the converging flow fronts meet.

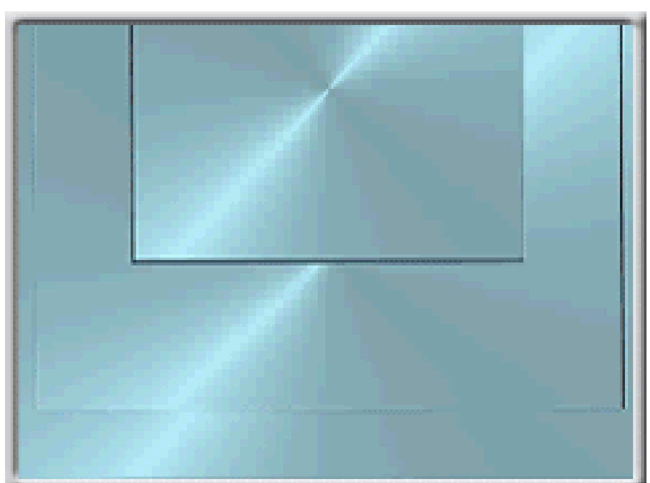


In the above diagram, the converging flow fronts (indicated by black arrows) meet. If the angle, θ is greater than 135° a meld line will form. If θ is less than 135° a weld line will form.

Weld lines

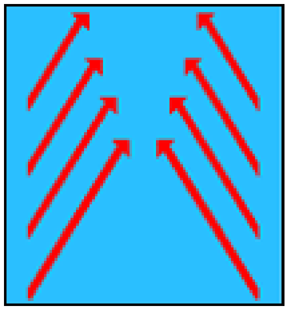
When a weld line forms, the thin frozen layers at the front of each flow path meet, melt, and then freeze again with the rest of the plastic. The orientation of the plastic at the weld is therefore perpendicular to the flow path. The following animation shows plastic filling a cavity. The weld line occurs where two flow fronts meet, and the polymer molecules are misaligned.

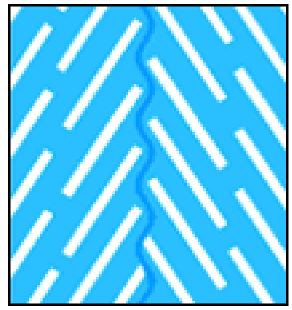
It is the sharp difference in molecular orientation at the weld which causes the significant decrease in strength at this point.



Meld lines

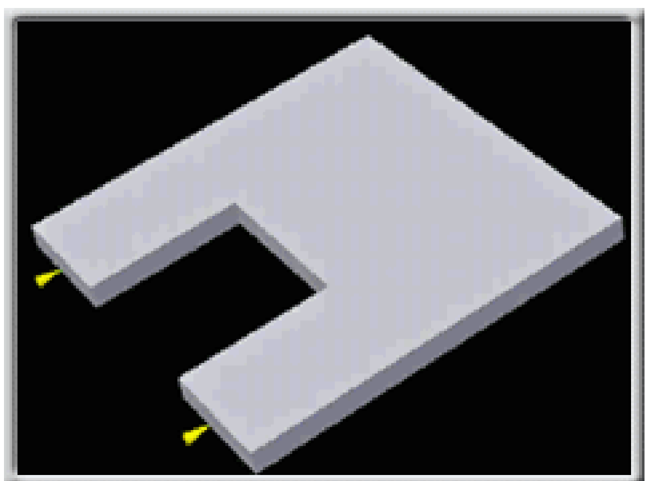
A meld line occurs when two flow fronts blend together at an oblique angle. The orientation of the plastic molecules is therefore more uniform than the orientation after a weld line has formed. The following diagram shows the length of a part where a meld line forms.





The red arrows show the direction of plastic flow. The white lines represent the orientation of the polymer molecules after the meld line has formed.

Meld lines are normally stronger than weld lines and are often much less visible.



NOTE: The term weld line is often used to mean both weld and meld lines.

Remedies

Weld and meld lines on a plastic part can cause structural problems and be visually unacceptable. (A line, notch and/or color change can appear.) Therefore weld and meld lines should be avoided if possible (when the cavity has unbalanced flow paths unnecessary weld and meld lines can occur).

If it is not possible to remove a weld/meld line, it should be positioned in the least sensitive area possible. Avoid weld lines in areas which need strength, or which need to appear smooth. This can be done by changing the polymer injection location or altering wall thicknesses to set up a different fill time. With a different fill time, flow fronts may meet at a different location and therefore the weld/meld line will move.

- Moving:
  - Change the gate positions.
  - Change the part thickness.
- Improving the quality:
  - Increase melt and mold temperature. This will allow the flow fronts to interfuse more.
  - Increase ram speed.
  - 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.

NOTE: The processing conditions help to determine the quality of the weld or meld line that has formed. A good weld occurs when the melt temperature is no lower than 20°C below the injection temperature.

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: Changes such as those above should only be made using a licensed Autodesk Moldflow Adviser or Autodesk Moldflow Insight product.

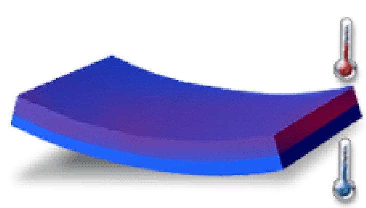
Warpage problems

Warpage occurs when there are variations of internal stresses in the material caused by a variation in shrinkage.

Warped parts may not be functional or visually acceptable.

Causes

- Non-uniform cooling. Temperature differences from one side of the mold to the other can lead to layers freezing and shrinking at different times and generating internal stresses.



- Inconsistent shrinkage. Resulting from:

1 Material variations such as property variations, varying moisture content, inconsistent melt and pigmentation;
2 Process conditions variations such as inconsistent packing and varying mold and melt temperatures;
3 Machine variations such as a damaged check ring and unstable controller.

Remedies

- Minimize differential shrinkage.
- Minimize orientation effects. Position gates for unidirectional flow, and modify part thickness.
- Change the part geometry. Add features such as stiffening ribs to the design. Change the part design to avoid thick sections and reduce the thickness of any features that intersect with the main surface.
- Use thinner wall sections with ribs. Thicken only those wall sections that require extra material for structural stability and that cannot be strengthened using another method.
- Change the material. Semi-crystalline have naturally higher shrinkage and hence are more prone to warpage.

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