

Injection-Molding Problems in Engineering Thermoplastics

Causes and Solutions

BASF

We create chemistry

TABLE OF CONTENTS

Click on the table of contents to jump to the desired section

	Table of contents	2
	Introduction	3
1	Weld line	4 - 7
2	Delamination	8 - 9
3	Diesel effect/Burning	10 - 11
4	Black specks	12 - 13
5	Sink marks	14 - 17
6	Demolding problems	18 - 21
7	Jetting	22 - 25
8	Gloss variations	26 - 27
9	Flash	28 - 29
10	Cold slug	30 - 33
11	Air entrapment	34 - 37
12	Voids	38 - 39
13	Dull spots	40 - 41
14	Record grooves effect	42 - 43
	Surface streaks	
15	Colored streaks	44 - 45
16	Moisture streaks	46 - 47
17	Streaks with reinforcements	48 - 51
18	Burning streaks	52 - 53
19	Tiger lines	54 - 55
20	Stress crack formation	56 - 57
21	Unmolten material	58 - 59
22	Short shot	60 - 61
23	Warpage	62 - 65
24	Plate-out / Mold deposit	66 - 67
	Troubleshooting table	68 - 69

Introduction

The demands on the quality of injection-molded components made from engineering thermoplastics have become ever more stringent in the past few years. This is due to the higher quality awareness among end consumers. In addition, technically sophisticated moldings are becoming more of a requirement in today's projects.

The complex interplay between the design of a molded part and the mold, between the properties of the chosen plastic and the diverse parameters of the injection-molding process, needs a lot of experience in order to achieve optimum results, especially if the aim is a short-notice remedy for processing errors which are frequently identified in the final stage of a project.

This guide provides support for technicians; it systematically lists errors which may occur when producing molded parts from engineering plastics. It also provides solutions which have proved their worth in application technology. Advice is also given as to which factors must be considered during the preliminary stages, i.e. machine recommendations, or when designing the component or producing the mold, in order to avoid errors before the production process is begun.

Note

This handbook contains advice on optimizing the processing temperatures. In general, it should be ensured that the processing temperatures do not exceed or fall below the limits of the melt and mold temperatures. In particular, exceeding the maximum melt temperature can cause molecular breakdown or cross-linking of the thermoplastic (for example, in the case of Ultrason®) and/or can cause the formation of gases and vapors.

BACK TO OVERVIEW

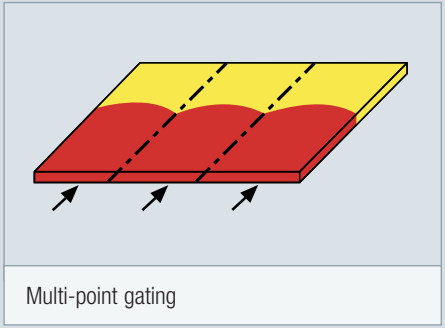
01 WELD LINE

Description

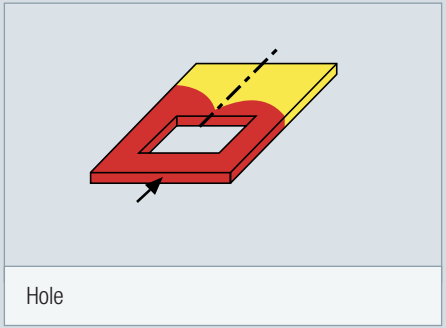
A scratch-like or notch-like line where the two melt flows meet during filling. This effect is particularly pronounced in dark or transparent plastic parts with polished surfaces, in parts with glassfiber reinforcement or in plastics with effect pigments. If the temperatures and pressures are too low in this area, the colder and stiffer melt skin on the edges no longer comes to lie against the mold surface and a visible notch is produced on the surface. In conjunction with poor flow fronts that are too cold, this can also lead to a deterioration in the mechanical properties. A weld line is often unavoidable. Its appearance can be improved by adjusting parameters. In addition a weld line can be moved to an area where it is less visible. Therefore, it is recommended to do a moldflow study prior to the mold being built.

Causes	Recommendations
Two or more melt fronts meet, having an effect on the appearance and usually also on the mechanical properties of the molded part.	PROCESSING CHANGES • Optimize the switch-over point to the holding pressure: shortly before the volumetric filling has been reached (approximately 98 % of the filling level), change over to holding pressure. • Improvements can often be achieved, for instance, by increasing the injection speed. However, if venting problems are encountered in the area of the weld line, the injection speed should be reduced. • Increase the melt temperature. • Clean the venting channels.
	MOLD-RELATED SOLUTIONS • By moving the gate and, if possible, by changing the wall thicknesses, the path of the polymer melt in the cavity and thus the position of the weld line can be influenced. Since this is associated with changes to the mold, the effectiveness of the measure should be checked by first carrying out a moldflow study. • Improve the venting in the affected areas of the mold to help filling.

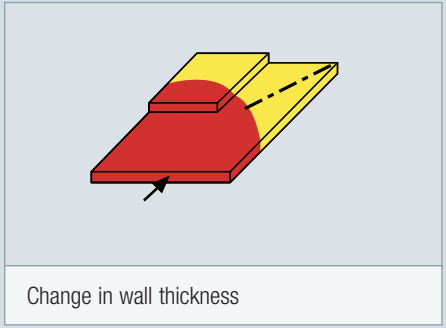
BACK TO OVERVIEW



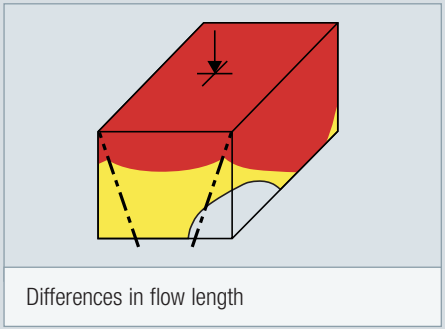
Multi-point gating



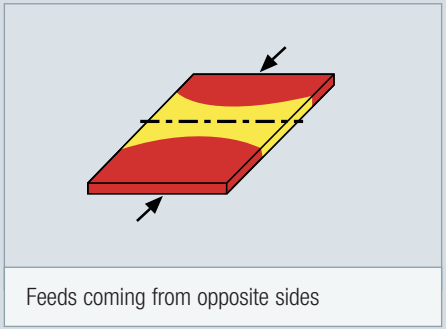
Hole



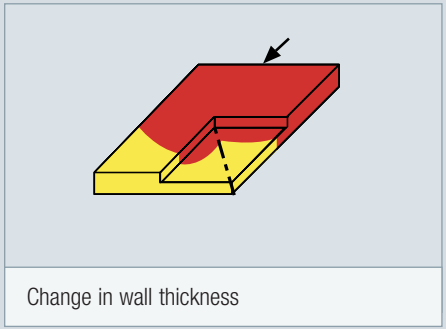
Change in wall thickness



Differences in flow length



Feeds coming from opposite sides



Change in wall thickness

Various geometric causes of weld lines

BACK TO OVERVIEW

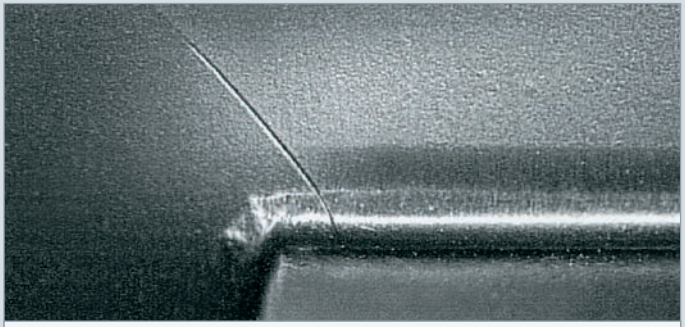


Weld lines after holes

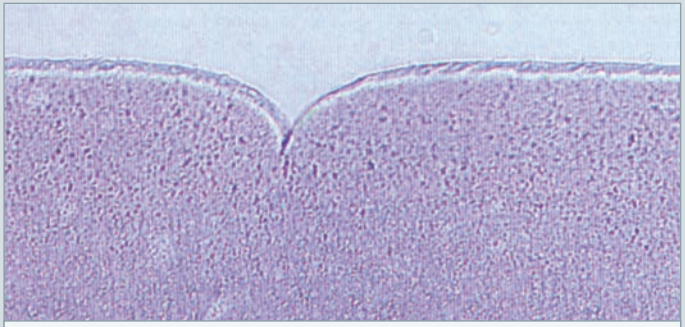




BACK TO OVERVIEW

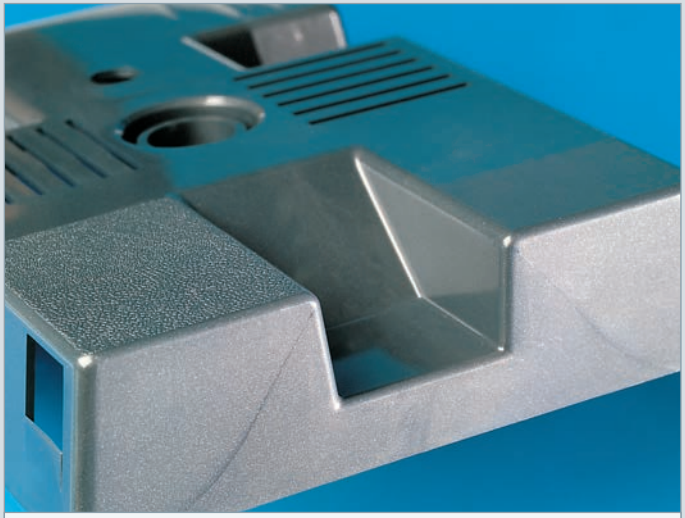


Vertical illumination, magnification 11:1

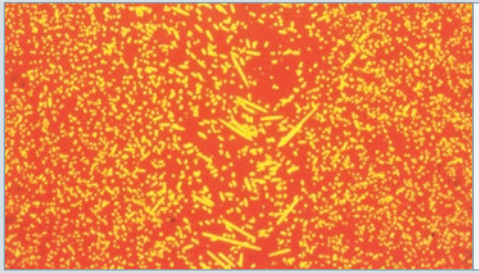


Transmitted light, polarized, magnification 560:1

Notches in a weld line



Weld line with metallic effect pigments



Orientation of glass fibers at the weld line

BACK TO OVERVIEW

Description
Delamination in plastic injection-molded parts refers to the peeling of surface layers. The cause of this inadequate bond between the polymer layers is excessive shear in the relatively cold melt in combination with intense cooling of the mold (cold mold). In the case of semi-crystalline thermoplastics, this can lead to the formation of layers having different crystal structures, while in amorphous thermoplastics, this can lead to de-mixing of the melt-additive-pigment mixture.

Causes	Recommendations
The layer formation is caused by excessive shear during the filling of the mold.	• Increase the melt temperature. • Reduce the injection speed. • Reduce the pressure loss in the sprue and the runners by enlarging the cross section. • Change the gate type, for example, a film gate instead of a pin gate. • Increase the wall thickness of the molded part.
The machine and possibly the hot runner are contaminated with foreign material.	• Thoroughly clean the machine and the hot runner prior to processing.
An incompatible masterbatch is used.	• Check whether the masterbatch is appropriate for the plastic being used. If necessary, change to a masterbatch with a suitable carrier.
Air becomes entrapped during the metering. Thin skins that can be pulled off are present on the surface of the molded part. This problem is encountered mainly with unreinforced, semi-crystalline thermoplastics.	• Raise the back pressure while the material is being fed. • Check whether the maximum permissible metering stroke has been exceeded; if necessary, use a machine with a larger screw. • Use a screw with an optimized design, for example, a shallow-flighted three-zone screw.

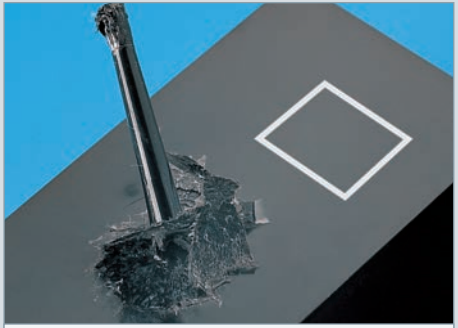
BACK TO OVERVIEW



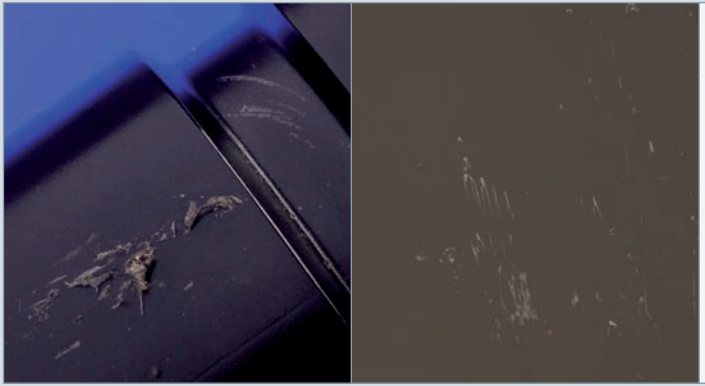
Delamination caused by an injection speed that is too high (mineral-filled Ultramid®)



Delamination due to an injection speed that is too high (Ultramid T)



Delamination due to material contamination



Delamination caused by entrapped air during metering (unreinforced Ultramid)

BACK TO OVERVIEW

Description
Locally black discolorations of the molded part occur in the area of the weld lines, at the end of the flow path or of individual areas of the molded part (for example, a rib). Occasionally, the molded part has not been completely filled at these places. It can also be the case that changes appear on the mold surface (plate-out or corrosion). The reason is that the injected plastic melt pushes air ahead of itself in the cavity, compressing it. If the air cannot escape through parting lines, ejectors or special venting inserts, it becomes so hot that it causes burnt spots on the plastic.

Causes	Recommendations
The mold is filled at an injection speed that is too high.	• Reduce the injection speed. It might already be sufficient to lower the speed only at the end of the filling phase (select a graduated injection profile).
The channels for venting the mold are contaminated or clogged by plate-out.	• Clean the mold venting system and improve the vents.
The air cannot escape through the parting surface of the mold.	• If the burning occurs close to the parting surface, reduce the clamping force of the machine.
The mold is insufficiently vented.	• Check the venting channels in terms of their position and effectiveness. If applicable, provide additional venting channels in the parting surfaces and/or install core pins in the mold at the places where melt flows meet.
Several melt flows meet.	• Analyze the filling situation. Adjust the flow pattern by optimizing the gate position or by providing flow promoters in such a way that the air can escape through the parting surfaces of the mold.

BACK TO OVERVIEW







Diesel effect in a rib due to a lack of venting





1. Venting improved by means of core pins

2. Lack of venting



The flow pattern shows the weld lines of several melt flows.

BACK TO OVERVIEW

04 BLACK SPECKS

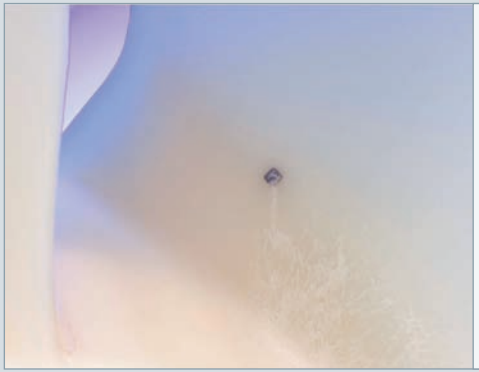
Description
Impurities or thermal damage to the polymer cause dark inclusions that become visible on the surface of the molded part or inside the molded part in the case of transparent plastics.

Causes	Recommendations
The material is contaminated with foreign matter.	• Reduce the time during which the granules are exposed to the open. • Check the condition of the material conveying system for possible leakages, dirt deposits or wear and tear. • Thoroughly clean the material conveying system, the dryer and the machine prior to any material change.
The temperature of the melt is too high.	• Check the melt temperature and, if necessary, lower the temperature of the barrel and of the hot runner. • Reduce the screw speed and the back pressure.
The material degrades in so-called dead spots in the plasticizing unit and/or in the hot runner.	• Check the design of the plasticizing unit (non-return valve, machine head, machine nozzle) and of the hot runner system and eliminate all dead spots.
Metallic particles come loose from the plasticizing unit and/or from the mold because of wear and tear.	• Check the condition of the plasticizing unit and/or of the mold for wear and tear or partial detachment of any coating that might be present; replace the affected components.
The residence time of the melt in the barrel and/or in the hot runner is too long.	• Check the residence time in the plasticizing unit, if applicable, change to a smaller barrel or use a screw with a shallow-flighted geometry. • Reduce the residence time in the hot runner, for instance, by means of a shorter cycle time or by reducing the cross sections of the flow channels.
The selected heating and/or cooling times are too long or the periods when the machine is at a standstill are too long.	• Heat up or cool down the cylinder in two stages (Ultrason®). • During interruptions, frequently purge out and/or reduce the temperature.

BACK TO OVERVIEW



Black specks



Black specks caused by foreign granules



Deposits from a hot runner following a standstill of the machine (Ultrason)



Impurities in a bottle due to dead spots in the plasticizing unit

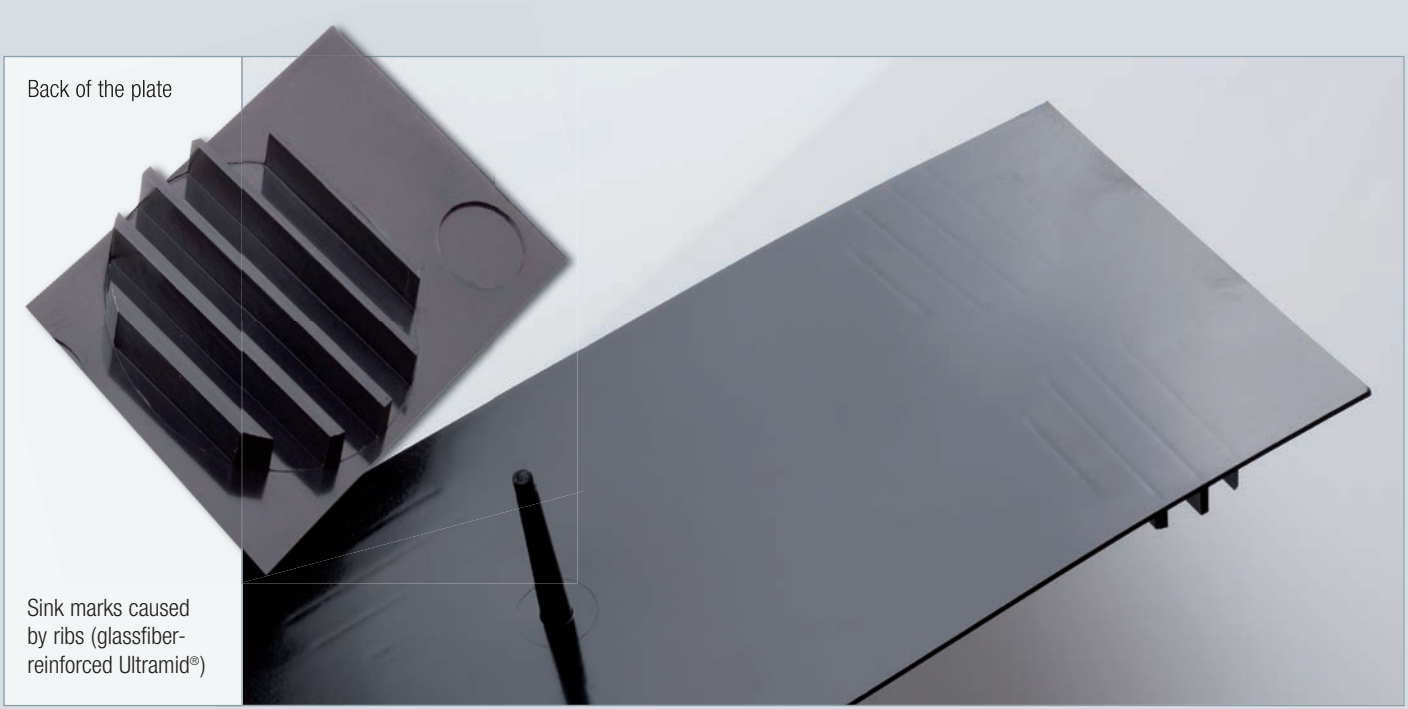
BACK TO OVERVIEW

Description

The term sink mark refers to indentations in the surface of the molded part. Sink marks occur mainly where there is an increase in a wall section. This causes a local increase in the volume shrinkage, which pulls the surface layer inwards. Sink marks sometimes only occur after ejection from the mold when the hot centre of the polymer heats up the edge layers that have already cooled off, causing them to yield. Sometimes they can only be recognized by the gloss difference in comparison to the surrounding areas.

Causes	Recommendations
The volume contraction during the cooling phase is not sufficiently compensated for by the holding pressure.	• Increase the holding pressure. • Increase the holding pressure time: first, check the gate open time by means of weight or pressure measurements. Starting with a short holding pressure time, raise this value incrementally until no more changes in the weight or in the cavity pressure can be detected.Select the optimal holding pressure time so that it is longer than the gate open time. • Reduce the melt temperature. • Decrease the mold temperature.
The design of the molded part is not appropriate for processing this plastic (e.g. material sections with an increase in wall thickness, sudden changes in the wall thickness along the flow path).	• Avoid sudden changes in the wall thickness and material sections, do not use excessively large radii for ribs; a rib thickness that is half the wall thickness is advantageous. • Change the wall thicknesses of the molded part so as to achieve even pressure transfer along each flow path. Avoid thin sections along a flow path.
There is no melt cushion.	• Lengthen the metering stroke and adjust the change-over point as necessary. • Check the sealing function of the non-return valve and, if necessary replace it. Check for barrel wear.
A large pressure loss has occurred in the machine nozzle and/or in the gating system.	• Enlarge the flow cross sections of the runner and gating system.
The molded part is gated into a thin wall.	• Always where possible, gate or inject the molded part in the area where the wall section is the greatest.

BACK TO OVERVIEW



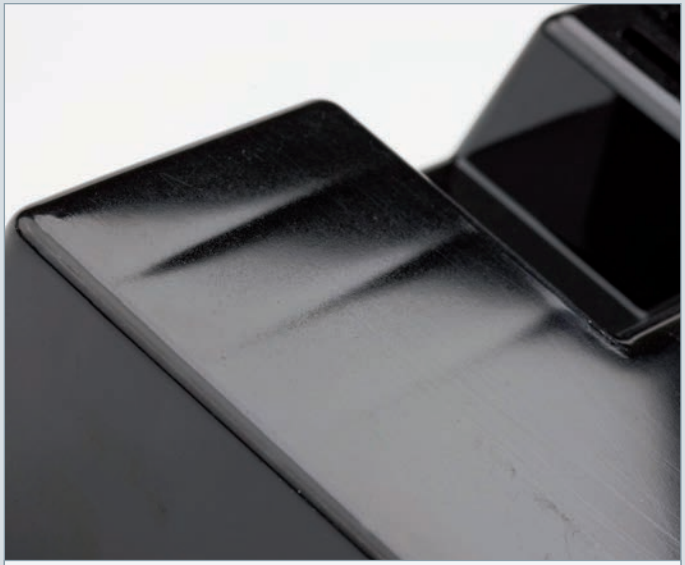
Back of the plate

Sink marks caused by ribs (glassfiber-reinforced Ultramid®)

BACK TO OVERVIEW

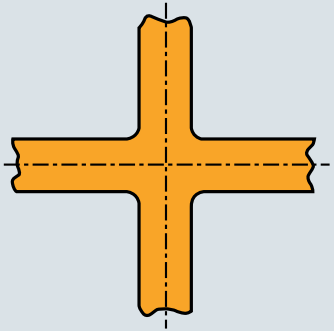


Sink mark in a section with an increase of wall thickness: the design of the screw-in mandrel is not appropriate for processing this plastic

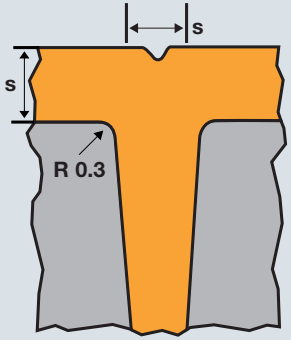


Sink marks caused by ribs

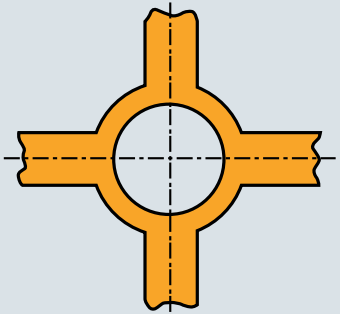
BACK TO OVERVIEW



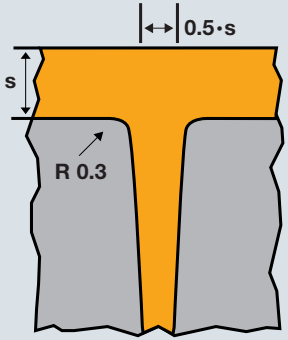
unfavorable



unfavorable



favorable



favorable

Avoiding sink marks (voids) by designing the rib connection so as to be appropriate for processing this plastic

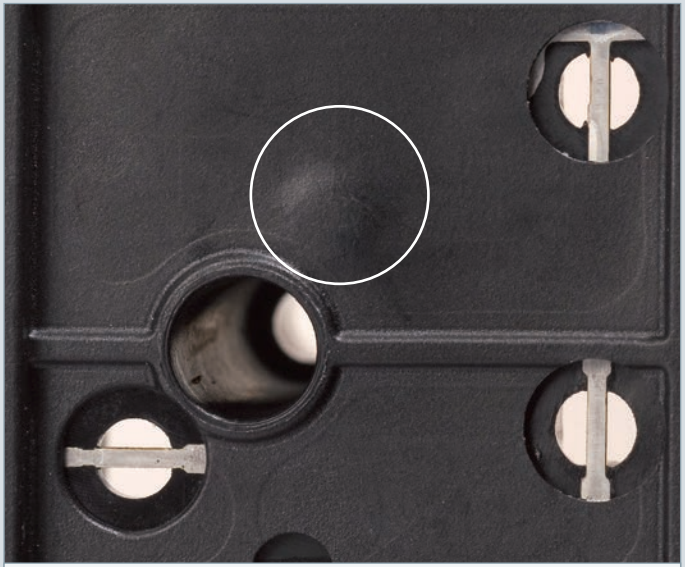
Design of ribs

BACK TO OVERVIEW

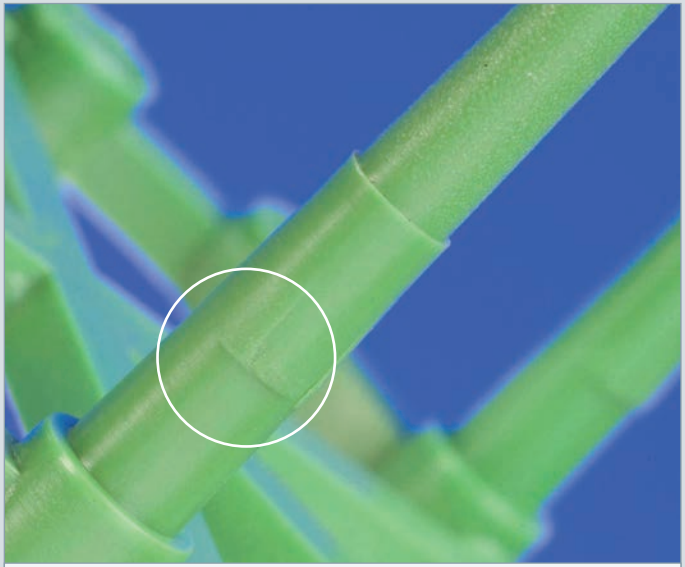
Description
The plastic part is deformed by forces that are too high or that are applied unfavorably during ejection. After the part has been removed from the mold, the ejectors leave distinct markings on the surface of the molded part as indentations or elevations, as differences in the gloss or as whitish discolorations (stress-whitening). Demolding problems can cause cracks or fractures of the material.

Causes	Recommendations
The molded part and/or the gating system have undercuts or scratches.	• Blend any undercuts. • Improve the surface quality of the mold. Polish it in the demolding direction.
The demolding system has too few ejector pins, they are incorrectly positioned or their surfaces are too small.	• Optimize the geometry of the ejector system.
The selected drafts are too small or the surface structure finish (texture) used does not match the draft.	• Increase the draft angle.
The mold is overpacked during the mold filling procedure due to excessive holding pressure.	• Adjust the switch-over point in such a way that, shortly before the volumetric mold filling level has been reached (filling level of approximately 98 %), a change is made over to holding pressure. • Reduce the holding pressure, if applicable, establish a decreasing holding pressure profile.
Owing to excessive shrinkage, the material shrinks too much towards the core, thus making the removal from the mold more difficult.	• Raise the holding pressure and, if applicable, employ a holding pressure profile.
The cooling time selected is too short.	• Prolong the cooling time.
The actual mold temperature is too low.	• Increase the mold temperature.
A vacuum is created between the molded part and the mold surface during the removal from the mold.	• Improve the venting of the mold inserts by incorporating channels, parting lines or special inserts into the mold through which the air can flow between the molding and the cavity, as the mold opens.

BACK TO OVERVIEW



Ejector markings caused by the cooling time being too short



The sprue gets stuck to the cavity plate because of an undercut.

BACK TO OVERVIEW

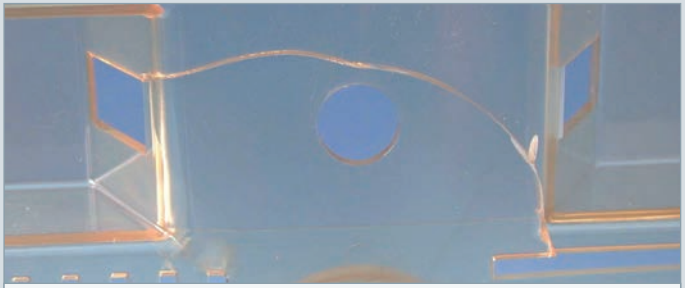


The ejectors puncture the molded part because the cooling time is too short.

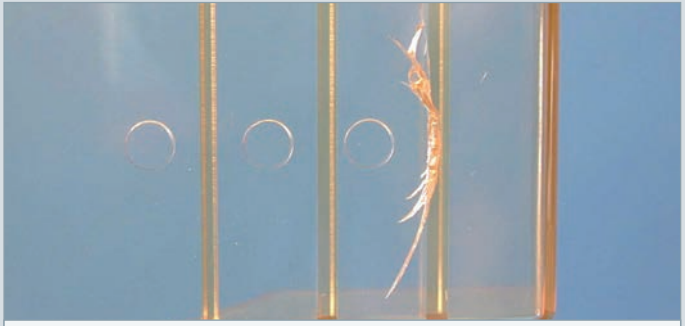


Mold-removal problems caused by overpacking (holding pressure is too high)

BACK TO OVERVIEW

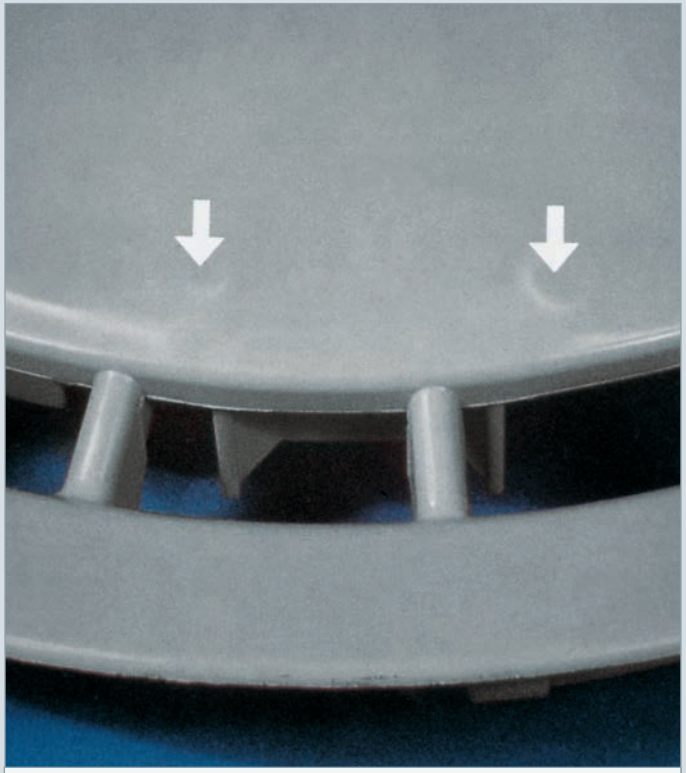


Corner with a sharp edge



Rib without draft, asymmetrical arrangement of the ejectors

Cracks occurring when molded parts made of Ultrason® are removed from the mold



Ejector markings – remedied by increasing the ejector diameters

BACK TO OVERVIEW

Description

Starting from the gate, a snake-like, often rough or matt strand of material appears on the surface of the plastic molded part. This occurs when, due to the very high flow rate during the transition to a large cross section, there is nothing to disturb the melt. Instead, the polymer melt squirts into the free cavity in the form of a jet, a process in which coincidental contact with the mold wall (brief adhesion) leads to crumpling. Since the surface of the strand has cooled off, it no longer bonds properly to the subsequent melt, which can lead not only to a poor appearance but also to reduced strength.

Causes	Recommendations
The position selected for the gate is incorrect. The gate design is not suitable.	• Change the gate position in such a way that the emerging polymer melt strikes an obstacle immediately beyond the gate. • Enlarge the cross section of the gate. • Round off the transition area between the gate and the molded part.
Unfavorable process conditions during the filling procedure.	• Reduce the injection speed. • Increase the melt temperature.
The design of the molded part is not appropriate for processing this plastic (for example, sudden changes in the wall thickness along the flow path).	• Change the design of the molding so as to prevent sudden changes in the wall thickness.

BACK TO OVERVIEW



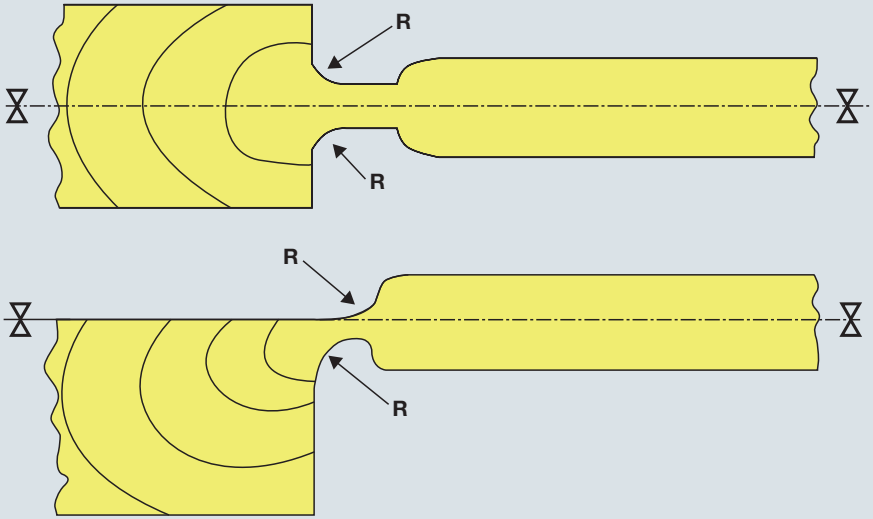
Jetting due to the unfavorable position of the gate



BACK TO OVERVIEW



Jetting on a sleeve caused by three-fold gating through a pin gate

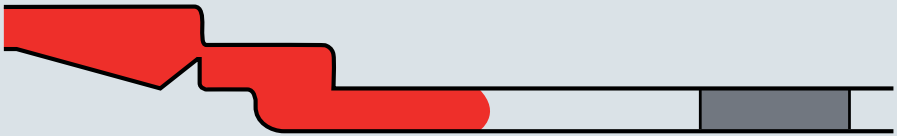


Ensure proper radii in the transition area from the gate to the molded part

BACK TO OVERVIEW

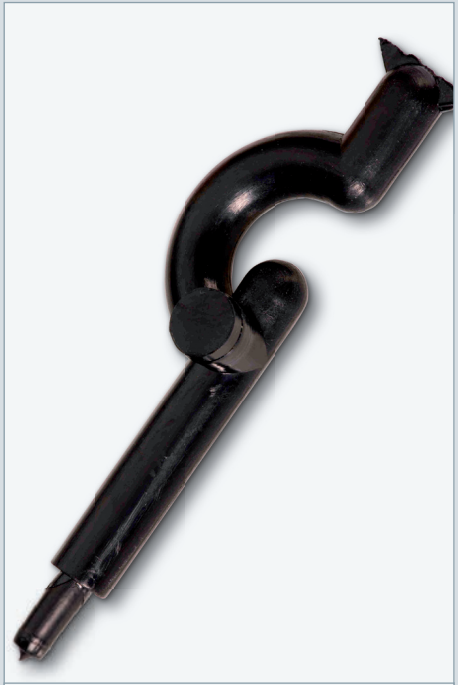


unfavorable



favorable

Gate design



Melt deflection in order to prevent jetting
(injection via the parting surface)

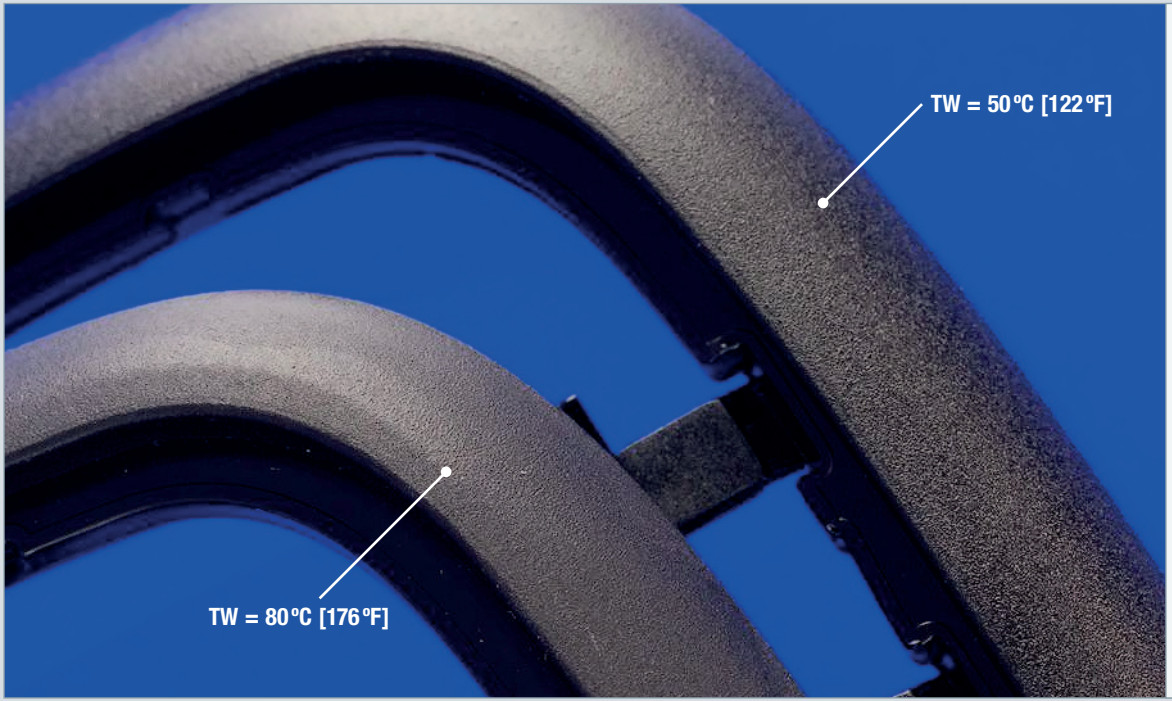
BACK TO OVERVIEW

Description

The surface of the plastic molded part shows different levels of gloss at different places, even though the cavity surfaces are the same. Generally speaking, the intensity of the gloss depends on how good the surface of the cavity is. In the case of structured mold surfaces, a well-designed mold usually yields a part with a more matt appearance since the rays of light are reflected diffusely, that is to say, undirected, off the many rough irregularities. In contrast, in the case of polished mold surfaces, a good impression usually brings about higher gloss. Key parameters are those adjusting variables which affect the formation of the solidified outer or peripheral plastic layer and its pressing against the wall of the mold. In addition, the polymer melt should be as uniform as possible.

Causes	Recommendations
Sudden changes in the wall thickness caused by ribs or by mounting bosses. These sink marks lead to differences in the gloss.	• Prevent material accumulations and sudden changes in the wall thickness; as much as possible, gate the molded part in the area where the wall is the thickest.
Differences in the gloss are visible in the area of weld lines, for example, due to a changed orientation of the melt and to changed flow conditions.	• Check whether, by modifying the gate position or by using flow promoters or flow retarders, the position of the weld lines can be changed to places where they do not interfere; a filling simulation is helpful in such a case.
The temperature of the mold wall, the temperature of the melt and the injection speed as well as the change-over to holding pressure have not been well-chosen.	• Increase the mold temperature. • Increase the melt temperature. • Optimize the injection speed. • Optimize the switch-over point in such a way that, shortly before the volumetric mold filling level has been reached (filling level of approximately 98 %), a change is made to holding pressure.
The holding pressure value and the holding pressure time are inadequate.	• Increase the holding pressure. • Check the holding pressure time. The holding pressure should be maintained until the sprue has frozen. The right holding pressure time can be found by weighing the part (incremental increase of the holding pressure time until no more weight increase occurs).
The homogeneity of the melt is insufficient, for instance, due to unfavorable processing conditions.	• Raise the back pressure and/or the screw speed. • Use a screw with a mixing section. • Use a shallow-flighted screw. • Install an injection unit with a screw having a larger diameter.

BACK TO OVERVIEW



Influence of the mold temperature (TW) on the gloss (glassfiber-reinforced Ultramid®)

BACK TO OVERVIEW

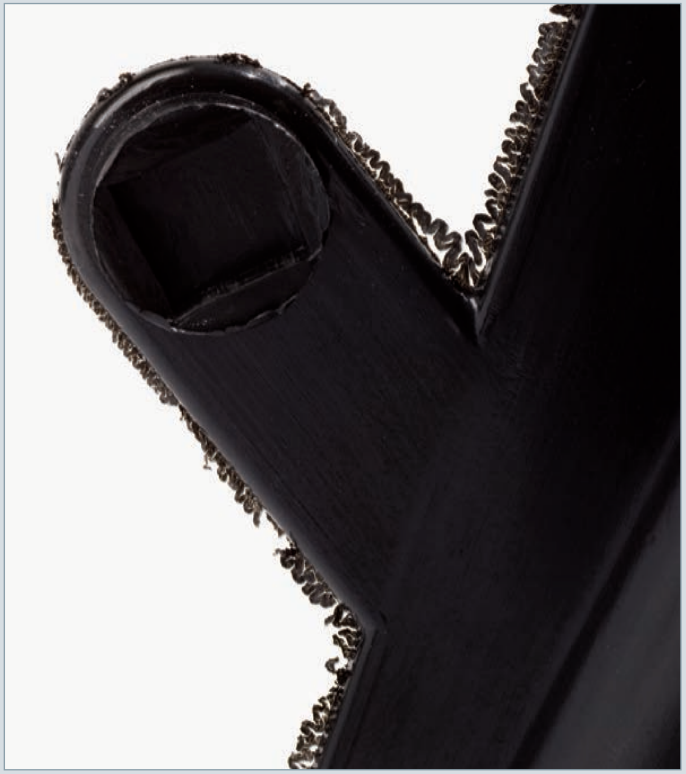
Description
The term flash refers to fine excess of material projecting from the plastic molded part. Flash occurs when the thermoplastic melt penetrates into gaps and joints located in the parting surfaces of the mold.

Causes	Recommendations
The clamping force of the machine is set too low or the clamping force of the selected machine is insufficient.	• Increase the clamping force. • Use a machine with greater clamping force.
The fit tolerance of the two mold halves or the fit tolerance of splits in the mold is too great.	• Adjust the fit tolerances of the mold parts involved.
The sealing surfaces in the mold parting surface are damaged.	• Repair the damage in the parting surface.
The cavity pressure is too high during the filling and holding pressure phases.	• Increase the temperature of the thermoplastic melt. • Adjust the change-over position to occur earlier. • Reduce the injection speed, especially at the end of the mold filling phase.
The viscosity of the thermoplastic melt is too low because the selected melt temperature is too high or because of material degradation. The material degradation can result, if the melt temperature is too high, if the residence time is too long or if moist material is being processed.	• Lower the melt temperature. • Reduce the residence time of the melt in the plasticizing unit and in the hot runner. • Dry the material.
The mold is deforming under filling pressure.	• Strengthen the mold construction.

BACK TO OVERVIEW







Flash

BACK TO OVERVIEW

10 COLD SLUG

Description

Due to unfavorable temperature conditions, the plastic will solidify in the machine nozzle or in the hot runner prior to the mold being filled. This will be injected with the next shot. Especially in the case of thin-walled or transparent parts, this can be recognized by the presence of markings near the gate in the form of a comet's tail or by the presence of locally concentrated streaks. If the slug does not melt, it can clog the flow channel to such an extent that the melt has to divide up, creating a weld line. In extreme cases, a cold slug can block off the gate, preventing the mold cavity from being completely filled. Moreover, the mechanical properties are detrimentally affected if the cold slug does not fuse properly with the rest of the melt.

Causes	Recommendations
The temperature of the machine nozzle is too low.	• Increase the nozzle temperature.
The machine nozzle is unheated or the output of the band heaters is too low.	• Heat the machine nozzle with additional heater bands. • Raise the output of the band heaters or install several heater bands on the machine nozzle.
The material drips out of the machine nozzle or out of the hot runner, cools off and is injected into the cavity at the time of the next shot.	• After plasticizing decompress the melt by retracting the screw a few millimeters. • Reduce the back pressure. • Lower the temperature of the machine nozzle. • Use a needle shut-off mechanism for the machine nozzle or for the hot runner nozzle. • Incorporate a cold slug area (catcher) within the sprue/runner.
The bore of the machine nozzle does not have any draft.	• Modify the bore of the machine nozzle with a reverse draft.
The tip of the tunnel gate breaks off when the part is removed from the mold and ends up in the cavity at the time of the next shot.	• Modify the angle of the gate in a way that is appropriate for the material. • Do not use a tunnel gate for brittle plastics.

BACK TO OVERVIEW

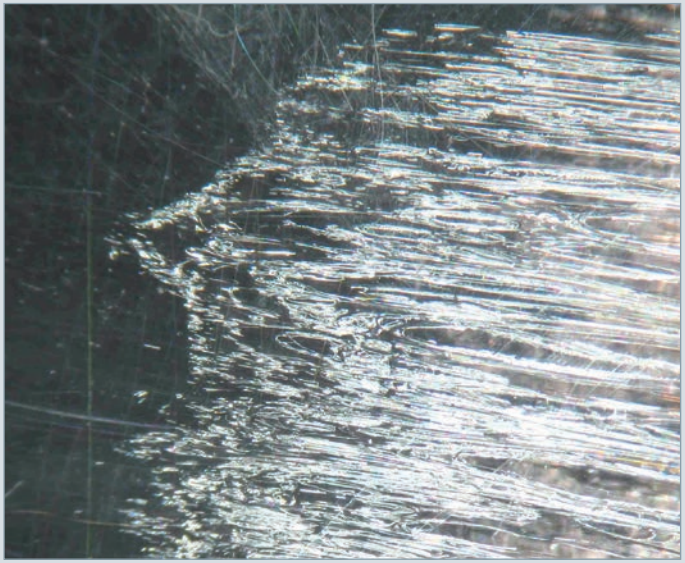


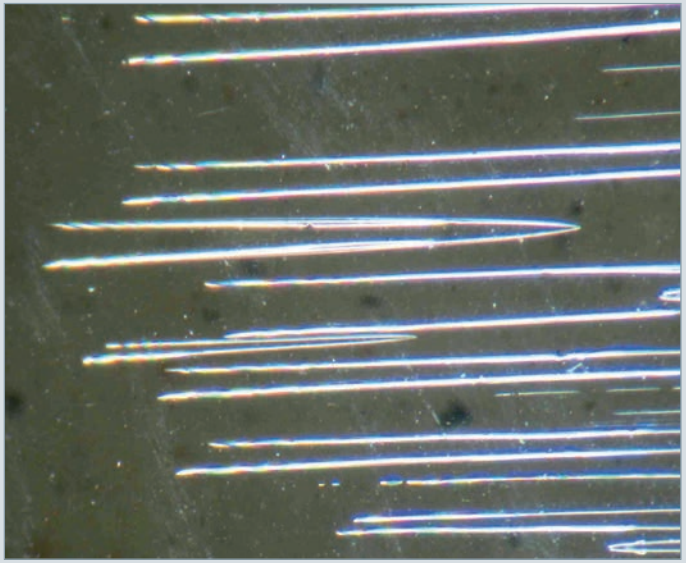
Cold slug from the bore of a machine nozzle that has cooled and broken off (glassfiber-reinforced Ultradur®)



Cold slug in a molded part made of glassfiber-reinforced Ultramid®

BACK TO OVERVIEW





Streaks in Ultrason® caused by cold slug

BACK TO OVERVIEW



Surface blemish caused by a gate that has broken off (unreinforced Ultradur)



Cold slug catcher in a gate with a deflection in order to prevent jetting (injection through the parting surface)

BACK TO OVERVIEW

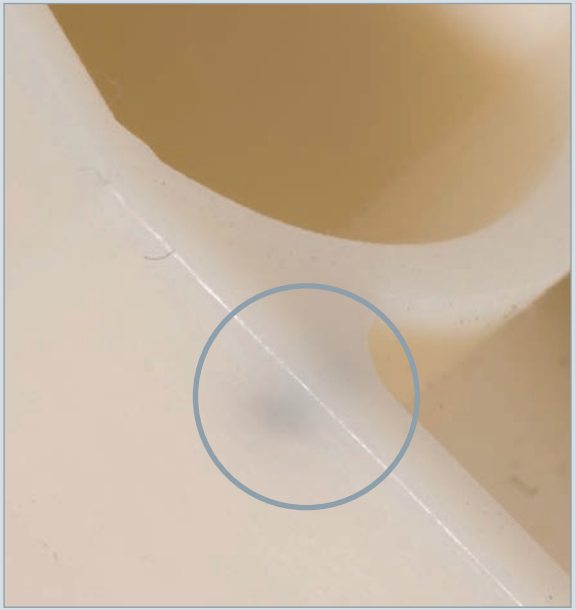
11 AIR ENTRAPMENT

Description

The appearance of this problem varies, depending on the cause of the air entrapment. If the polymer melt flows around the air, the flaw pattern can range from incomplete filling to blemishes on the surface and even to charred spots as a result of diesel effect. If the air is entrapped in the mold cavity by the thermoplastic melt, bubbles are formed. The bubble structure varies from small to large and often reveals the flow direction. In contrast to voids, the bubbles can be close to the outside wall.

Causes	Recommendations
During the mold filling, air is entrapped due to the molded part design and gating.	• Change the filling pattern by adding flow promoters and/or flow retarders. • Change the position of the gates. (In both cases it is recommended that the filling pattern be optimized by means of a moldflow study.)
After the plasticizing, air present in front of the screw or in the hot runner is entrapped in the melt because the screw decompression is too intense or too fast.	• Shorten the path of the screw decompression. • Decompress at a reduced screw retraction speed.
During the plasticizing of the material, air is entrapped via the screw because of processing conditions or because a screw that is not appropriate for the material is being used.	• Increase the back pressure when the material is being metered in. • Lower the temperature in the feed zone. • Check whether the maximum permissible metering stroke has been exceeded; if necessary, use a machine with a larger screw diameter. • Use a screw with an optimized design, for example, a shallow-flighted three-zone screw or a screw with a shorter feed zone.

BACK TO OVERVIEW



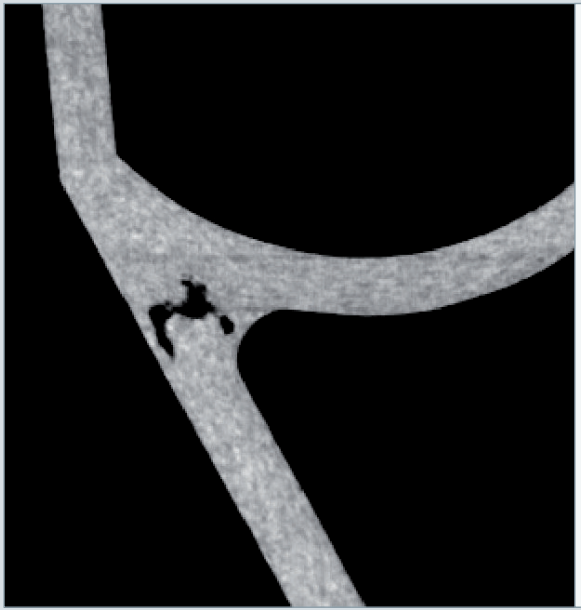
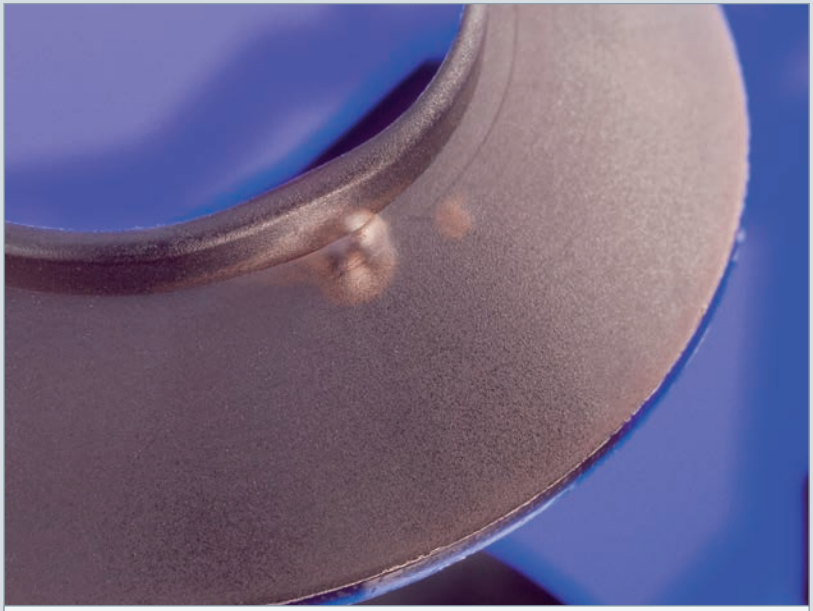


Image of a section of a molded part, taken by 3D computer tomography

Air entrapment in a housing made of glassfiber-reinforced Ultradur®

BACK TO OVERVIEW

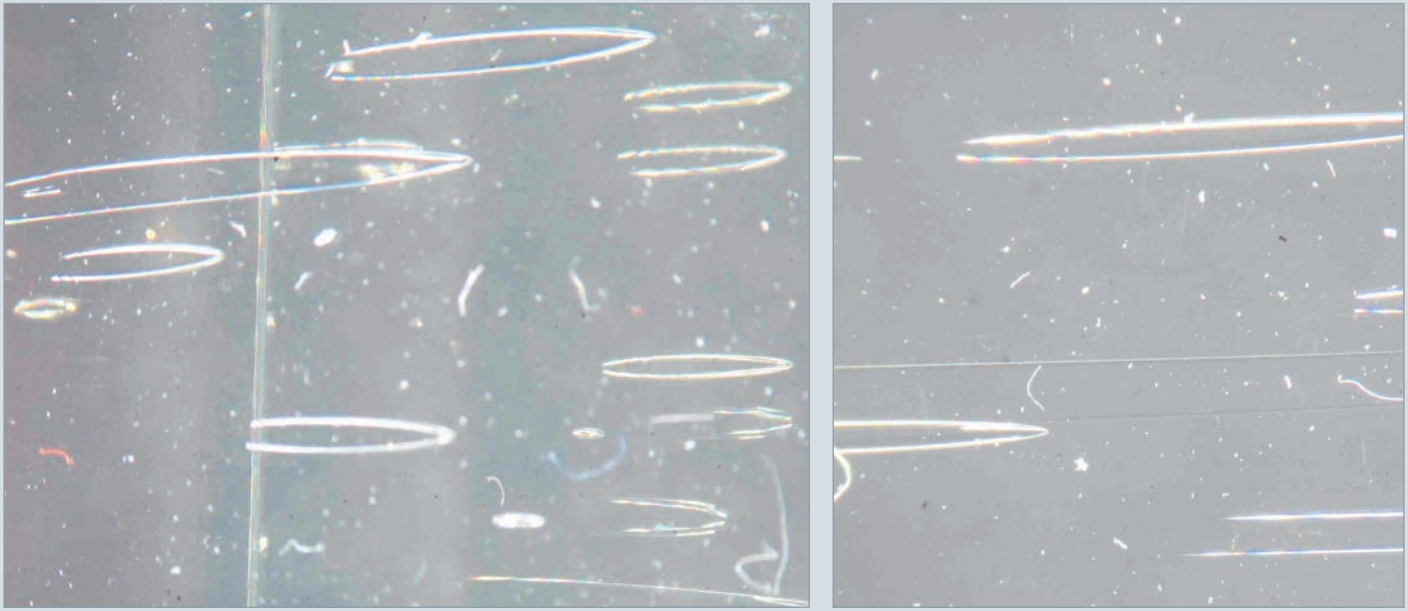


Air entrapment caused by entrapped air during the metering (unreinforced Ultramid®)



Air entrapment caused by an unfavorable molded part design

BACK TO OVERVIEW



Streaks caused by entrapped air (Ultrason®)

BACK TO OVERVIEW

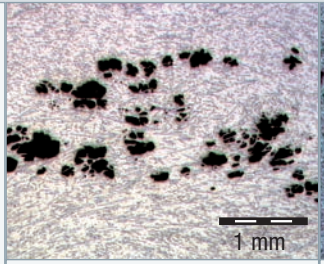
Description

Hollow spaces in the form of microcells or bubbles are formed inside the plastic molded part during or after the cooling phase. In contrast to gas bubbles, voids always occur in the molten center of the plastic (normally the middle of the wall) and in areas with thick sections. Drilling into the part while holding it under water shows whether water is penetrating (voids) or not (gas bubbles). As is the case with sink marks, voids are also formed in areas of high volume shrinkage due to inadequate packing of the material. Voids are formed instead of sink marks whenever the edge layers have solidified to such an extent that they no longer yield to the contraction forces when the material cools off.

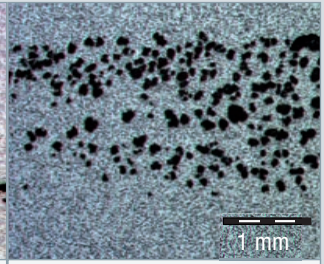
Causes	Recommendations
The volume contraction during the cooling phase is not sufficiently compensated for by the holding pressure.	• Raise the holding pressure. • Prolong the holding pressure time: first of all, the gate open time is ascertained by means of weight or pressure measurements. Starting with a short holding pressure time, raise this value incrementally until no more changes in the weight or in the cavity pressure can be detected. Select the optimal holding pressure time so that it is somewhat longer than the gate open time. • Reduce the melt temperature. • Increase the mold temperature. In this case, optimize the holding pressure time by checking the gate sealing point.
The design of the molded part is not appropriate for processing this plastic (e.g. presence of material accumulations, sudden changes in the wall thickness along the flow path).	• Avoid sudden changes in the wall thickness and material sections, do not use excessively large radii for ribs; a rib thickness that is half the basic wall thickness is advantageous. • Change the wall thicknesses of the molded part so as to achieve proper pressure transfer along each flow path. Avoid thin sections along a flow path.
There is no melt cushion.	• Lengthen the metering stroke and adjust the change-over point. • Check the sealing function of the non-return valve, if necessary, replace it; check the barrel for wear.
A large pressure loss has occurred in the machine nozzle and/or in the gating system. The molded part is gated into a thin wall.	• Enlarge the flow cross sections of the runner and gate; use round diameter runners where possible. • Try to gate or inject into the molded part in an area where the wall has the thickest section.

BACK TO OVERVIEW

Polished section, optical light micro-scopic images with vertical illumination



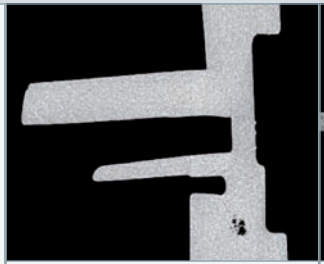
Voids resulting from excessively short holding pressure time



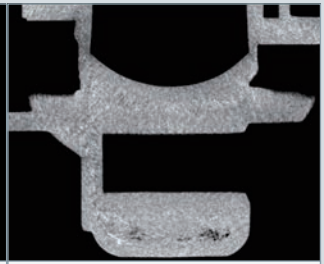
Voids in the area of a sudden change in the wall thickness

Voids in molded parts made of glassfiber-reinforced Ultramid®

Images taken by 3D computer tomography

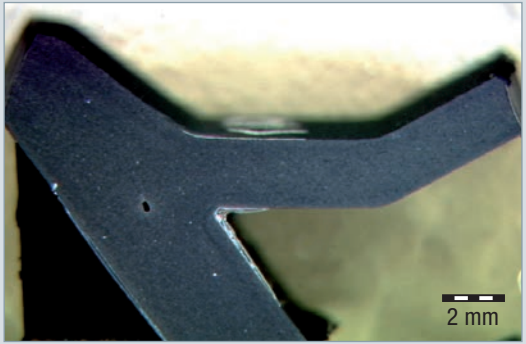


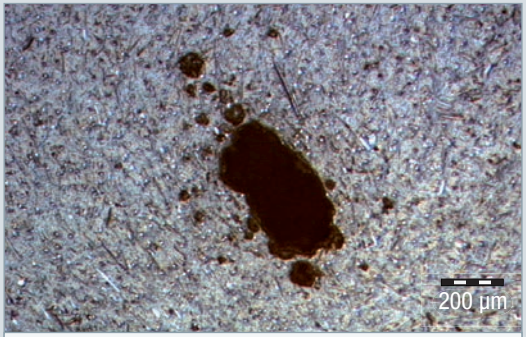
Bearing element made of Ultraform®



Housing made of glassfiber-reinforced Ultradur®

Voids caused by material accumulation





Detail from image above, polished section, optical light microscopic image with vertical illumination

Voids caused by material accumulation in an engine cover made of reinforced Ultramid

BACK TO OVERVIEW

13 DULL SPOTS

Description
Halo-like dull spots are often formed in the area of gates; sometimes they also occur behind cross section transitions. The causes for this are the high shear stresses at these places as well as the diminished adhesion of the plastic to the wall due to the cross section transitions.

Causes	Recommendations
The selected injection speed is too high.	• Optimize the injection speed, set a graduated profile (slow – fast).
The gate is not optimally configured.	• Optimize the gate. To the extent possible, the gate should be designed so as to be rounded, having an adequate cross section.
The temperature selected for the melt is too low.	• Increase the melt temperature.
There is an abrupt cross section transition.	• Optimize the design of the molded part; ensure a gradual cross section transition.
The molded part is gated at an unsuitable place.	• Move the feed point to a less critical area or gate the molded part via a separate runner located outside the visible area.

BACK TO OVERVIEW



Dull spots in the area of a film gate



Dull spots caused by a cross section transition



Dull spots in the area of the gate caused by high shear

BACK TO OVERVIEW

14 RECORD GROOVES EFFECT

Description

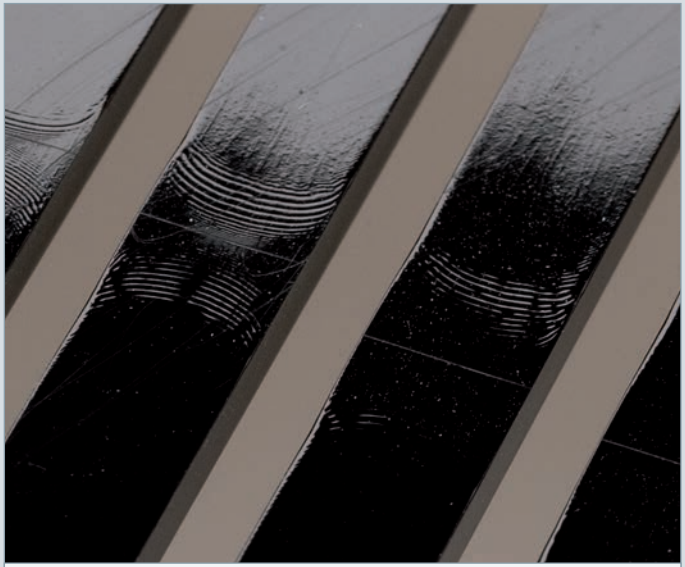
The record grooves effect refers to markings that are formed perpendicular to the flow direction of the polymer melt, appearing as a concentric or parallel fluted structure on the surface of the molded part. The reason for this can be that the convex melt front cools down excessively due to a reduced flow rate (possibly only at times). The solidified plastic edge layer can become so thick that it extends all the way into areas of the melt front that are not yet touching the mold wall. These areas can no longer be pressed against the mold wall over their entire surface, even as the filling continues and thus the pressure rises.

Causes	Recommendations
The selected mold temperature, melt temperature and injection speed are too low.	• Increase the temperature of the mold surface. • Select a higher injection speed. • Increase the melt temperature.
The design of the molded part is not appropriate for processing this plastic (for instance, thicker walls at the end of the flow path or a sudden change in the wall thickness).	• Optimize the geometry of the molded part and avoid sudden changes in the wall thickness.
A large pressure loss has occurred in the gating system.	• Increase the gate diameters.

BACK TO OVERVIEW



Record grooves effect because the melt temperature and the injection speed are too low



Record grooves effect at the end of the flow path because the melt viscosity is too high

BACK TO OVERVIEW

Description

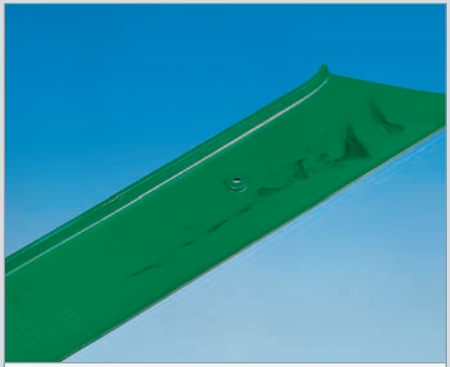
Color differences on the surface of injection-molded parts are referred to as colored streaks. They can occur over a large surface area and can be near the gate or away from the gate. They frequently appear in areas of weld lines and occasionally behind sharp edges. The reason for this is an irregular distribution of the pigments (either compounded colored material or masterbatch). This can give rise to the formation of agglomerations or accumulations of pigments (agglomerates).

Causes	Recommendations
During self-coloring	
Unfavorable processing conditions have been selected.	• Raise the back pressure. • Lower the screw speed. • Increase the melt temperature.
The selected plasticizing unit is unsuitable for self-coloring due to inadequate dispersing action.	• Use a plasticizing unit with a longer screw. • Install a plasticizing unit with a mixing section. In actual practice, the BASF mixing ring has proven to work well for self-coloring. • Use a shallow-flighted screw instead of a standard screw. Shallow-flighted screws allow a more gentle melting in conjunction with a greater homogeneity of the melt.
Processing of colored materials	
The pigments are damaged due to an excessively high thermal and/or mechanical load during the plasticizing or the filling procedure.	• Check the melt temperature and, if necessary, lower it. • Select a lower injection speed. • Reduce the screw speed and the back pressure.
Foreign material accumulates in dead spots in the plasticizing unit and/or in the hot runner.	• Thoroughly clean the plasticizing unit and the hot runner.

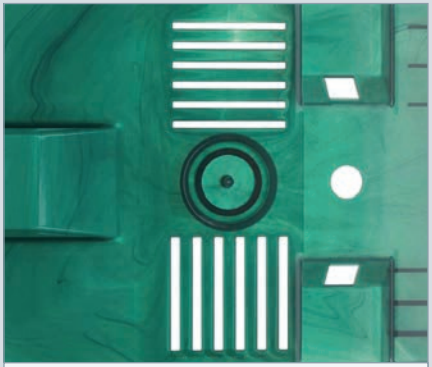
BACK TO OVERVIEW



Colored streaks caused by the use of an unsuitable masterbatch



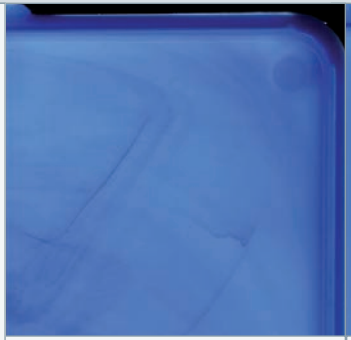
Colored streaks due to impurities in the hot runner



Colored streaks caused by the use of a screw with an unsuitable geometry

Avoidance of colored streaks by using mixing sections

Ultraform® N2320 003 + 0.5 % batch blue



Non-return valve



BASF mixing ring



BASF mixing ring

BACK TO OVERVIEW

Description
Moisture streaks are usually elongated, parabolic streaks that occur on the surfaces of molded parts. Their points always face in the direction of flow. Melt emerging from the machine nozzle can foam and display bubbles.

Causes	Recommendations
The material has too much residual moisture. When it is melted, steam vapor bubbles are formed that burst while the mold is being filled and come to lie on the surface of the molded part.	• Measure the moisture content of the granules; if applicable, dry the material to a value that is lower than the limit value indicated for the product. Check the dryer unit is working correctly.
The mold, whose temperature is controlled with water, is leaking into the cavity.	• Repair the mold tool of all leaks. • Select negative pressure on mold heater controls for a temporary solution.
The temperature in the area of the feed hopper temperature control is too low, condensation water is formed.	• Increase the cylinder temperature below the feed hopper.

BACK TO OVERVIEW





Moisture streaks in Ultrason®

moist granules

Processing of moist Ultradur® (purged melt cake)



dry granules (moisture content of 0.01 %)

moist granules (moisture content of 0.12 %), melt cake with bubbles



dried granules

BACK TO OVERVIEW

Description
Streaks in glassfiber-reinforced or mineral-filled materials can appear in the form of rough spotty and irregular surface areas but also as uniform surface blemishes whose shape partially matches the form of the flow front in this area of the molded part. Depending on the light incidence, the appearance can vary from matt to metallically glossy. Such streaks occur primarily after openings, diversions and at weld lines.

Causes	Recommendations
The injection speed is too low.	• Increase the injection speed; it might be practical to select a graduated injection profile.
The temperature of the mold surface is too low.	• Increase the mold temperature.
The selected melt temperature is not high enough.	• Increase the temperature of the melt, if necessary, also in the hot runner.
The orientation of the glass fibers changes in areas where there are material accumulations, for example, in areas with sudden changes in the wall thickness, in ribs or with mounting bosses.	• Check whether material accumulations and sudden changes in the wall thickness can be avoided.

BACK TO OVERVIEW

Glassfiber streaks influenced by the product



glassfiber-reinforced PA



glassfiber-reinforced Ultramid® High Speed

BACK TO OVERVIEW

Glassfiber streaks influenced by the product

glassfiber-reinforced PA



glassfiber-reinforced Ultramid Surface Improved (SI)





Glassfiber streaks caused by low mold temperature (glassfiber-reinforced Ultramid)

BACK TO OVERVIEW

Streaks in a molded part made of mineral-filled Ultramid – optimization of the processing conditions

TM = melt temperature

v = injection speed



TM = 260 °C [500 °F], v = low



TM = 280 °C [536 °F], v = high

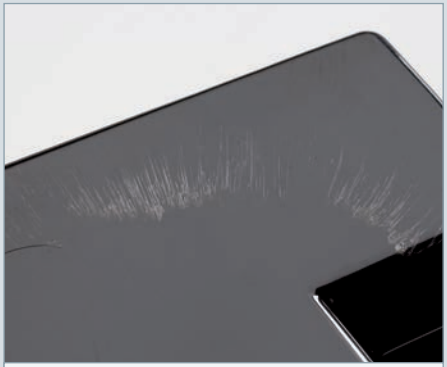
BACK TO OVERVIEW

Description

Charred streaks occur in the form of silver, light-brown to very dark discolorations on the surface of the molded part. The cause for this is severe thermal damage to the polymer melt. Gaseous bubbles are formed that reach the surface of the molded part during the injection phase. Charred streaks can also occur at and around the gate area as well as irregularly on the surface of the molded part. The appearance of the streaks allows a preliminary classification of the type of damage. Light-brown to very dark discolorations often indicate severe thermal damage to the plastic melt caused by oxidation or degradation. Silver streaks normally result from excessive friction, which is usually locally limited, for instance, in the case of small flow channel cross sections or sudden flow changes around obstacles.

Causes	Recommendations
The melt is damaged by an excessively high temperature.	• Measure the melt temperature and check whether it falls within the permissible processing window and, if applicable, lower the temperature of the melt in the barrel and in the hot runner. • Reduce the screw speed and/or the back pressure.
The melt is damaged by an excessively long residence time.	• Check whether a machine has been stopped. • Check the residence time in the barrel, if applicable, change to a smaller barrel or use a shallow-flighted screw. • Reduce the residence time in the hot runner, for instance, by selecting a shorter cycle time or by reducing the channel diameters.
The streaks are caused by high shear during the mold filling procedure.	• Optimize the injection speed and profile, if necessary. • Check the flow channel cross sections of the gate, of the sprue and, if applicable, of the hot runner system and enlarge them.

BACK TO OVERVIEW



Burning streaks due to thermal degradation



Burning streaks due to high shear in the gating system

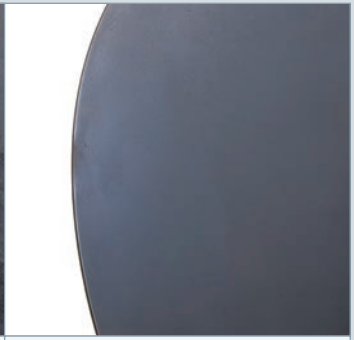


Streaks caused by thermal factors

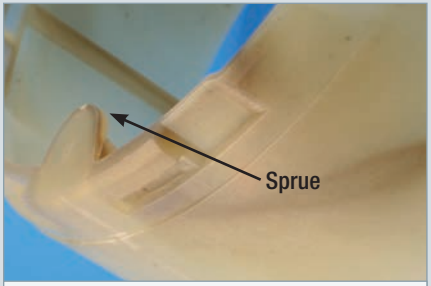
Burning streaks caused by exceeding the maximum limit for the melt temperature (TM) in the case of Ultramid® T



TM = 340 °C [644 °F]



TM = 320 °C [608 °F]



Burning streaks caused by excessively high shear in the gate (glassfiber-reinforced Ultrason®)

BACK TO OVERVIEW

19 TIGER LINES

Description
The term tiger lines refers to the alternating occurrence of glossy and matt stripes on the surface of the molded part perpendicular to the flow direction. They resemble the pattern on the coat of a tiger and are caused by a pulsating melt flow which occurs especially with thermoplastic multiphase systems (blends).

Causes	Recommendations
The cross sections of the gate, sprue and runners are too small.	• Increase the cross sections of the gating system and runners.
The selected cross section of the machine nozzle is too small.	• Install a machine nozzle with a larger bore diameter.
The thickness of the molded part wall is not sufficient.	• Check whether it is possible to increase the thickness of the wall of the molded part.
A large pressure loss occurs along the flow path.	• Increase the temperature of the melt.
The tool temperatures are too cold.	• Increase the tool temperature to the maximum recommended.

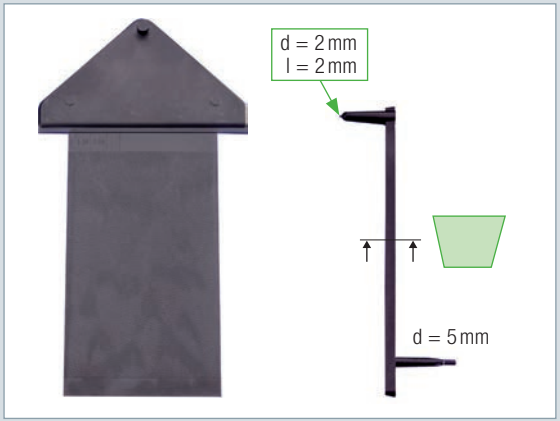
BACK TO OVERVIEW

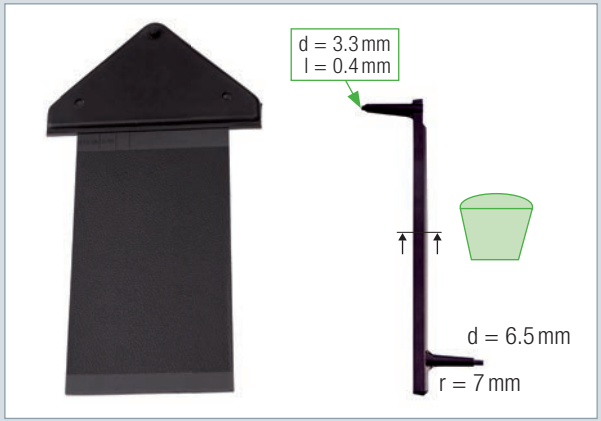


Tiger lines caused by a gate that is too small (glass-fiber-reinforced Ultramid®)



Tiger lines caused by unfavorable processing conditions (Ultraform®)





Improved molded part surface due to an improved gating system with less pressure loss

BACK TO OVERVIEW

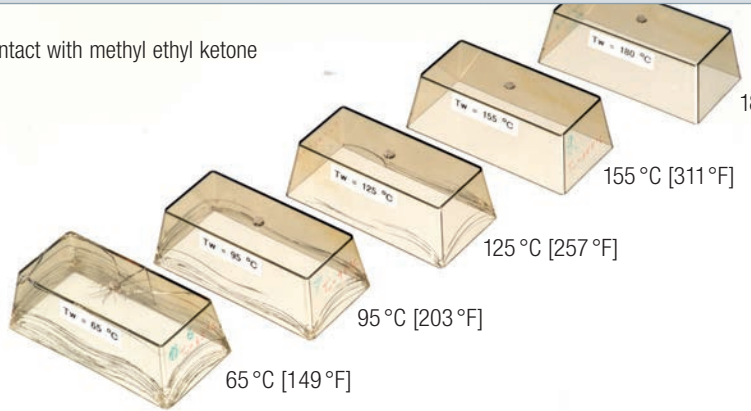
Description

Stress cracks are external or internal cracks caused by stresses that are less than the ultimate tensile stresses. Local internal stresses, among other things, are responsible for the formation of stress cracks. The crack formation is initiated either by external tensile stress and/or by the effect of aggressive solvents and chemicals. The magnitude of the internal stresses incorporated into the injection-molded part is influenced by the processing. Process-related internal stresses can be analyzed using solvents or chemicals, which initiate stress cracks.

Causes	Recommendations
The material is overloaded because the holding pressure is too high or the mold construction is not good enough.	• Reduce the back pressure, if applicable, set a decreasing back pressure profile. • Improve the mold construction.
The temperature selected for the mold is too low.	• Increase the mold temperature.
Cracks are formed during ejection owing to severe external forces because of the forced removal from undercuts from the mold. This is due to not enough draft angle and the number of ejectors being insufficient, because the ejectors are positioned incorrectly.	• Reduce the undercut or avoid it through design measures. • Improve the removal from the mold by means of larger drafts. • Optimize the ejector system.
The molecular weight of the selected product is too low; this has to be taken into consideration when agressive solvents or chemicals are used.	• Use a material with a higher molecular weight.

BACK TO OVERVIEW

After contact with methyl ethyl ketone



Processing window for the mold temperature: 140 °C to 180 °C [284 °F to 356 °F]

Influence of the mold temperature on stress cracking (unreinforced Ultrason®)



Stress cracks caused by overloading (holding pressure too high)



Stress cracks because of low mold temperature after media contact (Ultrason)

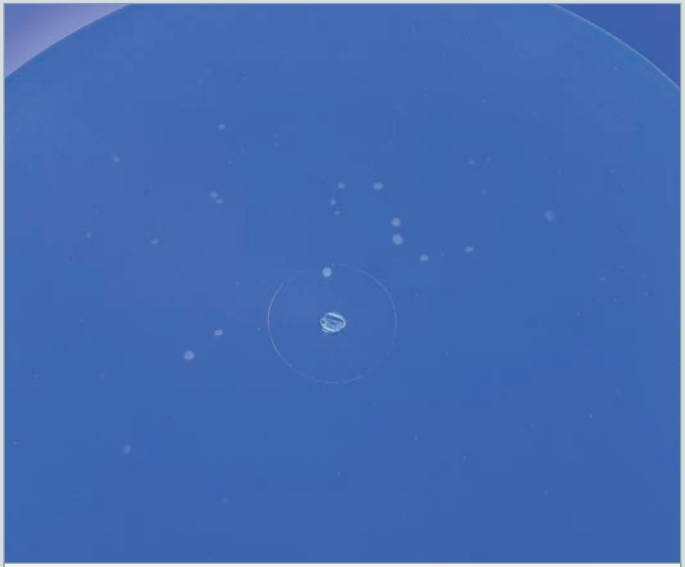
BACK TO OVERVIEW

21 UNMOLTEN MATERIAL

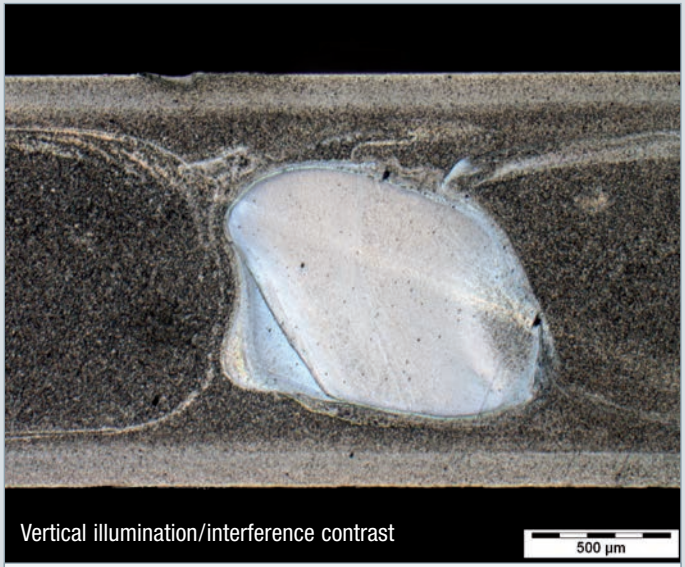
Description
Under unfavorable plasticizing conditions, unmolten material can pass through and end up in the molded part in the case of semi-crystalline thermoplastics. If the walls are thin, the unmolten material leaves a contour in the form of elevations on the surface. In the case of unreinforced, uncolored product bubbles having greater transparency can be seen.

Causes	Recommendations
Unfavorable plasticizing conditions have been selected.	• Select a horizontal temperature profile for the cylinder (instead of a rising profile). • Increase the melt temperature. • Raise the back pressure. • Pre-heat the granules.
The residence time of the material in the cylinder is too short.	• Use a machine having a larger screw diameter. • Use an injection cylinder with a longer screw.
The screw being used has an unsuitable geometry.	• Use a screw with a design that is appropriate for the material. Positive results have been obtained with shallow-flighted three-zone screws.

BACK TO OVERVIEW



Unmolten material due to unsuitable screw geometry (Ultraform®)



Unmolten material in a molded part made of Ultraform because the metering stroke is too large

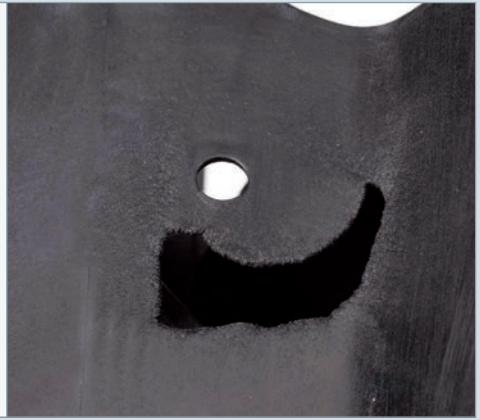
BACK TO OVERVIEW

Description
The plastic molded part is not completely formed, mainly in areas furthest from the gate or in areas with thin walls, since the mold has not completely been filled.

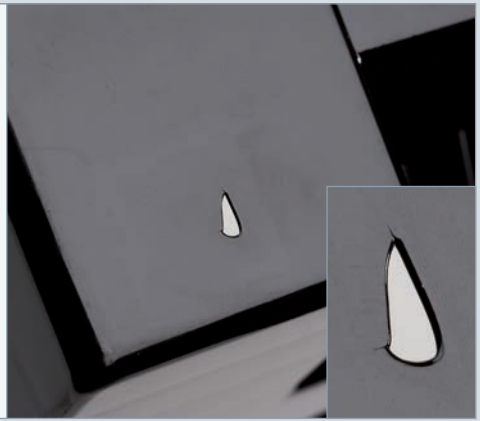
Causes	Recommendations
Not enough plastic melt was metered in.	• Increase the metering stroke.
There is no melt cushion.	• Check the function of the non-return valve.
The selected melt temperature and/or mold temperature are too low.	• Increase the melt temperature of the plastic melt and/or the mold temperature. The melt temperature should be increased first since it lowers the filling pressure. Increase the mold temperature, which, however, could also lengthen the cooling time.
The selected injection speed is too low.	• Increase the injection speed.
The mold is not adequately vented.	• Check the vents and improve the venting of the cavity if necessary.
The gating system has too much pressure loss.	• Enlarge the flow channel cross sections of the machine nozzle, of the runners and of the gates.
The molded part has been designed with a wall that is too thin or the selected flow path is too long.	• Increase the wall thickness of the molded part. • Change the gate position or increase the number of gates. • Introduce a flow leader, if possible.
Near the gate, there is a thin-walled area of the molded part that already freezes during the mold filling procedure.	• Change the gate position in such a way that the area with the thin wall is the last to be filled. • Introduce an additional runner.
The flowability of the selected plastic is insufficient, even after the process parameters have been optimized.	• Use a material that flows more easily.

BACK TO OVERVIEW

Incompletely filled molded part due to large pressure loss in the gate



Incompletely filled molded part







Incompletely filled molded parts

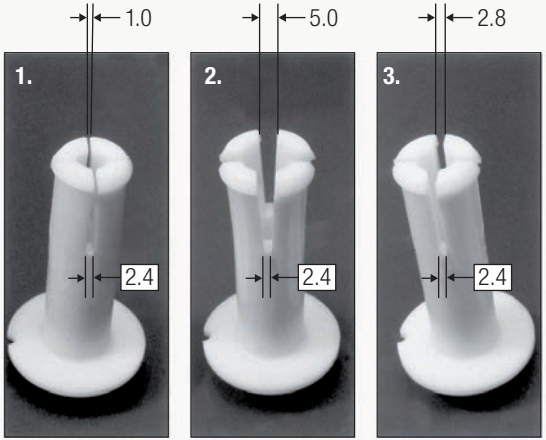
BACK TO OVERVIEW

Description

The plastic molded part deviates from the desired shape; it exhibits twisting, warping, wavy surfaces and angle deviations. The cause for this is the different shrinkage tendency (shrinkage potential) of the various areas of the molded part. The shrinkage differences are caused by different degrees of packing in different areas of the molded part as well as by different orientations.

Causes	Recommendations
The selected mold temperatures are unfavorable.	• Select different temperatures for the mold halves and sliding cores; this solution is often only helpful in the case of unreinforced thermoplastics.
The molded part has different wall thicknesses.	• Optimize the design of the molded part. Reduce sudden differences in the wall sections, blend out theses differences if possible.
The pressure gradient of the molded part is too high along the flow path.	• Gate the molded part in several places (only recommended in the case of unreinforced thermoplastics).
Since the selected cooling time is too short, the molded part is removed from the mold at too high a temperature; the molded part becomes deformed after being removed from the mold.	• Select a longer cooling time. • Check if the molding is not catching or being held with negative draft.
An unfavorable material is being used.	• Use low-warpage materials, e.g. blends with an amorphous phase. In such products, the difference between the lengthwise and the crosswise shrinkage – which causes the warpage – is smaller.
The glass fibers of glassfiber-reinforced thermoplastics are oriented predominantly in the flow direction. Warpage occurs if the orientation of the glass fibers changes from place to place. The causes for this are, for example, deflections in the flow, orientation effects at the end of the flow path, at the weld lines and at the gates.	Numerical simulation methods can be employed to predict the filling behavior and the orientation of the glass fibers. The following rules have proven advantageous for optimizing the warpage: • Gate the molded part so as to establish a uniform flow direction (= orientation). • Gate elongated molded parts in the lengthwise direction. Symmetrical parts prove to be the best. • Avoid ribs or walls perpendicular to the flow direction. • Position the end of the flow path in corners whenever possible. • Strive to create abutting weld lines whenever possible (keep the strength in mind). • Avoid weld lines on exposed webs and place them into the corners whenever possible.

BACK TO OVERVIEW

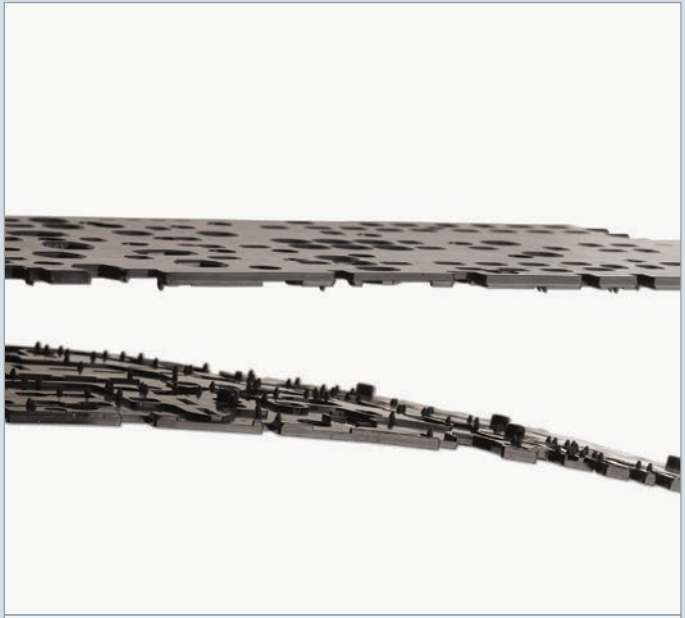


Holding mandrels made of Ultraform®

Influence of the mold temperature on the warpage

Mold temperature:

	Cavity	Core	Play
1.	80 °C [176 °F]	80 °C [176 °F]	- 1.4
2.	80 °C [176 °F]	30 °C [86 °F]	+2.6
3.	80 °C [176 °F]	50 °C [122 °F]	+0.4



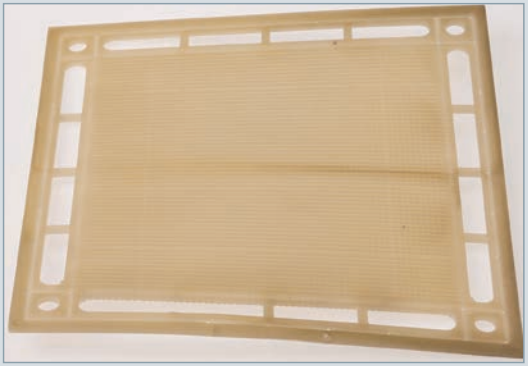
Insulating panel made of glassfiber-reinforced Ultradur®

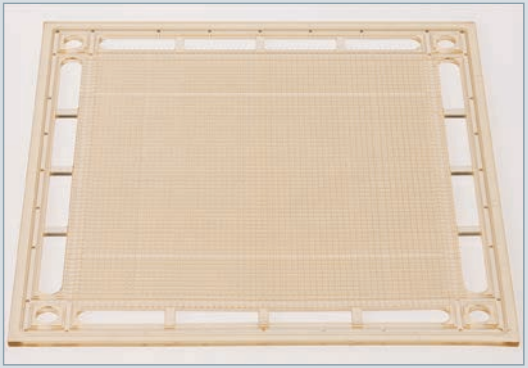
top: molded part geometry optimized by simulation
bottom: original situation

BACK TO OVERVIEW



Warpage of a power drill housing made of glassfiber-reinforced Ultramid®



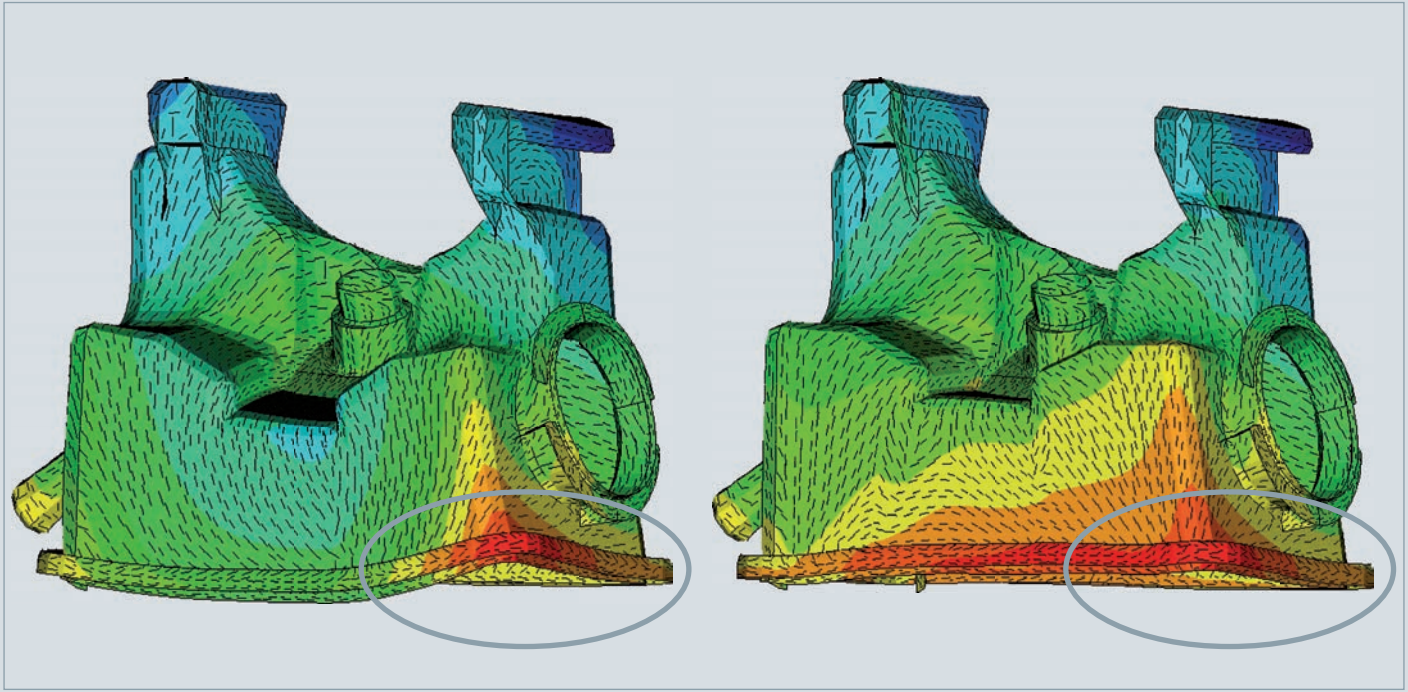


Warpage of a molded part made of Ultrason®

glassfiber-reinforced

unreinforced

BACK TO OVERVIEW



Simulation of the glass fiber orientation and warpage of a housing

left: initial situation, right: optimized geometry of the molded part

BACK TO OVERVIEW

24 PLATE-OUT / MOLD DEPOSIT

Description
This refers to impurities on the surface, in the parting surface and in the venting system of the mold. The plate-out can be breakdown products of the polymer, additives or flame retardants.

Causes	Recommendations
The melt temperature is too high.	• Use a needle thermometer to check whether the temperature of the melt falls within the recommended processing window as it emerges from the machine nozzle and, if applicable, also in the hot runner. If necessary, lower the temperature.
The injection speed is too high during the filling procedure.	• Reduce the speed of the screw. It might be sufficient to lower the speed only at the end of the filling phase (select a graduated injection profile).
The material has not been dried sufficiently.	• Measure the moisture content of the granules; if applicable, dry the material to a value that is lower than the limit value indicated for the product.
The mold is not adequately vented.	• Check the venting channels in terms of their position and effectiveness. If applicable, incorporate additional venting channels into the parting surfaces and/or incorporate core and ejector pins into places of the mold where the melts meet.
The residence time in the plasticizing unit and/or in the hot runner is too long.	• Check the residence time in the plasticizing unit; if applicable, switch to a smaller barrel or use a shallow-flighted screw. • Reduce the residence time in the hot runner, for instance, by setting a shorter cycle time or by reducing the flow channel diameters.
The melt is exposed to severe shear along the flow path.	• Check the design of the gate, sprue and runners and, if applicable, enlarge the flow cross section. Install a machine nozzle with a larger flow diameter.

BACK TO OVERVIEW

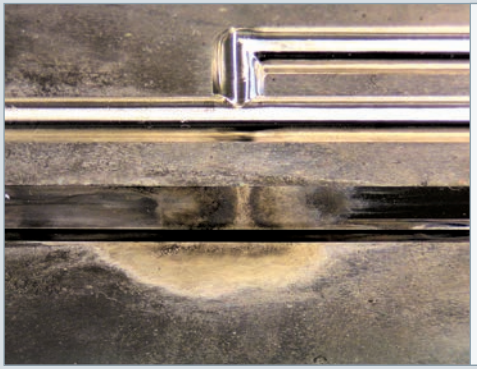
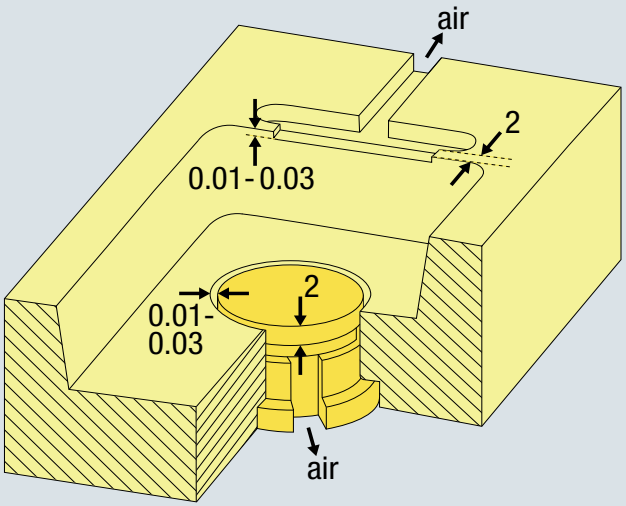


Plate-out/mold deposit in the area of a weld line due to a lack of venting



Plate-out/mold deposit at the end of the flow path due to a lack of venting



In the case of materials that flow easily select small venting slits of, for example, 0.01 mm to 0.02 mm, in order to avoid flash.

Recommendations for the design of the mold venting set-up

BACK TO OVERVIEW

TROUBLESHOOTING TABLE

PROBLEMS / SOLUTIONS	Melt temperature	Mold temperature	Cooling time	Injection speed	Holding pressure	Holding pressure time	Back pressure	Screw speed	Metering path	Screw retraction	Clamping force	Granule moisture	Contaminated granules	Mold design	Molded part design	Gate cross section	Screw geometry
Weld line	1	2		3													
Delamination	3			2									1				
Diesel effect	2			1							3						
Black specks	2						3						1				
Sink marks	3				1	2											
Demolding problems		3	1		2												
Colored streaks	3						1	2									
Moisture streaks												1					
Jetting	2			1												3	
Gloss variations	2	1			3												
Flash	3			1	2												
Cold slug							2			1				3			
Air entrapment							2			1				3			

BACK TO OVERVIEW

PROBLEMS / SOLUTIONS	Melt temperature	Mold temperature	Cooling time	Injection speed	Holding pressure	Holding pressure time	Back pressure	Screw speed	Metering path	Screw retraction	Clamping force	Granule moisture	Contaminated granules	Mold design	Molded part design	Gate cross section	Screw geometry
Voids	3				1	2											
Dull spots	2			1										3			
Record grooves effect	3	1		2													
Streaks (glass fiber, mineral)	3	2		1													
Tiger lines	1														3	2	
Stress crack formation		1			2	3											
Unmolten material	1		3				2										
Short shot	3			2					1								
Burning streaks	1			2				3									
Warpage		1	2												3		
Plate-out/ Mold deposit	2			1				3									

The effectiveness of each recommendation is indicated by the numbers, with 1 being the most effective.

= increase = reduce = change

BACK TO OVERVIEW

Note
The data contained in this publication are based on our current knowledge and experience. In view of the many factors that may affect processing and application of our product, these data do not relieve processors from carrying out own investigations and tests; neither do these data imply any guarantee of certain properties, nor the suitability of the product for a specific purpose. Any descriptions, drawings, photographs, data, proportions, weights etc. given herein may change without prior information and do not constitute the agreed contractual quality of the product. It is the responsibility of the recipient of our products to ensure that any proprietary rights and existing laws and legislation are observed. (October 2026)

If you have any technical questions about the products, please contact the Infopoint:



Please visit our websites:
www.plastics.basf.com
www.plastics.basf.de

® = registered trademark of BASF SE

PMK2620BE