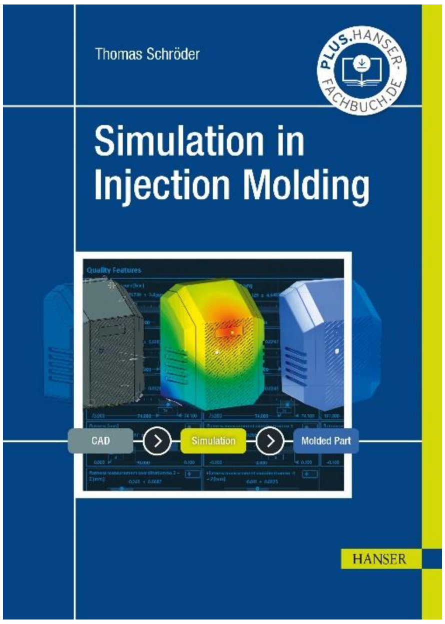
HANSER



Sample Pages

Simulation in Injection Molding

Thomas Schröder

Print-ISBN: 978-1-56990-916-4
E-Book-ISBN: 978-1-56990-932-4

For further information and order see

www.hanserpublications.com (in the Americas)

www.hanser-fachbuch.de (outside the Americas)

Contents

Preface

Simulation software programs are now a staple in the field of injection molding for the design of molded parts and molds. They assist designers early in the development of the molded part and mold design, thereby reducing the often time-consuming modification cycles. This shortens development time and lowers development costs. Additionally, the programs can enhance the quality of the molded parts and the efficiency of the injection molding process. Given the wide range of simulations that these programs can perform, the breadth of the simulation results is equally extensive. The challenge usually lies not in generating the results produced, as these programs are all very user-friendly, but in understanding and interpreting them. Only when this is the case can the user draw meaningful conclusions from them and suggest potential improvements. This is exactly where this book comes in. It begins with a general introduction to the injection molding process, meshing, and basic mathematics, followed by simple basic exercises. These are designed to provide the reader with an initial understanding of the simulation technique of how to interpret the results. Each chapter then explains the prerequisites for injection molding, the framework conditions and mathematical correlations, before conducting sample simulations out on real parts. For each specific simulation case, the results are interpreted and solutions developed. Finally, the aim is to use these simulations to make the reader aware of the essential points, which can vary extensively from part to part.

As the book contains numerous pictures and graphics and these are not always very easy to see due to their size, we have decided to provide the reader with all the pictures additionally in the form of image data. At *plus.hanser-fachbuch.de,* you can access the images using the code on the first page of the book.

Darmstadt 2024

Dr. Thomas Schröder

Foreword

For the most successful companies in plastic injection molding, simulating every part in advance has become the standard way of working. Whether you're a seasoned professional or still learning the ropes, simulation amplifies your skills, allowing you to identify and prevent process or part issues before any metal is machined. This includes issues like filling errors, warpage or excessive cycle times, among others. Another key advantage that is often overlooked is how simulation results and images bring clarity and focus to discussions with colleagues, superiors, and customers. By visualizing potential outcomes you can have a more constructive discussion, because your customers and colleagues will more quickly understand what you mean.

Naturally, mastering such a powerful tool takes some practice. You need to gain a deep understanding of how to represent reality as accurately and efficiently as possible if you want accurate results. But even the best simulation results are only as valuable as the insights you draw from them. Simulation is not just a prediction tool – it's a decision-making tool. That's why Prof. Dr. Thomas Schröder's focus on correctly interpreting simulation results is spot-on. His 30 years of expertise in plastics technology, distilled into this book, will save you from many potential missteps, and help you capture the full value of simulation.

Mastering the insights from this book can significantly advance your career. Given the pace at which engineering departments are digitalizing their ways of working, skilled simulation experts are increasingly in demand. By reading this book, you will not only enhance your professional knowledge but also your market value.

Ines Oud
CEO of injection molding simulation company **SIMCON** and Chairwoman of the German plastics industry association **Kunststoffland NRW**

The Author

Prof. Dr. Thomas Schröder teaches injection molding, rheology, mold technology and simulation technology at Darmstadt University of Applied Sciences (h_da Hochschule für Angewandte Wissenschaften), Germany, and is a member of EUt+ (European University of Technology). After studying mechanical engineering with a specialization in plastics technology at RWTH Aachen University, he completed his doctorate under Prof. Dr. Dr. h. c. Walter Michaeli on the subject of gas injection technology. After working for several years at a well-known plastics manufacturer, he moved to Krupp Corpoplast in Hamburg, where he was responsible for injection molding systems that produce preforms. Following this position, he moved to Netstal Maschinen AG in Näfels, Switzerland. There, he headed the application technology SPA of the injection molding machine manufacturer until he received a call to the University of Applied Sciences Darmstadt in 2001. He is also the author of the book *Rheologie der Kunststoffe* (Rheology of Plastics), the second edition of which was published by Hanser Verlag in 2020. Prof. Dr. Thomas Schröder is a member of the Institut für Kunststofftechnik Darmstadt ikd and chairman of the Gesellschaft zur Förderung technischen Nachwuchses GFTN e. V. (Society for the Promotion of Young Technical Talent) at Darmstadt University of Applied Sciences. Furthermore, he is a member of the PhD Centre for Sustainability Sciences and has the right to award doctorates, due to his numerous research projects, i. e., third-party funded projects in the field of injection molding, rheology, simulation, and tool technology. Prof. Dr. Thomas Schröder is also the managing partner of PlastSolutions Consulting GmbH, based in Mannheim.

Acknowledgments

This book could not have been written without the invaluable cooperation of many Bachelor's and Master's students. In this respect, I would first of all like to thank the many graduates whom I was able to supervise in various subject areas. I wish you all the best and much success in your professional career and also privately! In particular, I would like to thank the following people for their active participation in the book project: Aletta Berger, Sophie Dolata, Mary Göhler, Laura Gollan, Markus Hammermeister, Fabian Nebel, Bardo Palmberg, Bastian Paulsen and Maximilian Teich.

Furthermore, I would like to thank the company Simcon, which has always supported me with the software modules Cadmould and Varimos. Finally, I would like to thank the company Mold and Hotrunner Technology MHT, which also assisted me with the preparation of this book.

1 Basics of Injection Molding

1.1 Components of an Injection Molding Machine

Almost one third of all plastics are processed by injection molding. This master-molding technique is a discontinuous process. Injection mold machines are used for producing moldings from thermoplastics, elastomers and thermosetting plastics. These machines consist of several modules such as the machine bed, the plasticizing and injection unit, the clamping and locking unit, the temperature control unit as well as the mold and the control unit (Figure 1.1) [1].

The machine bed serves to accommodate the clamping, locking and injection units. In addition, pumps and tanks for hydraulics oil are built into the base of machines that need hydraulics. In fully electric injection molding machines, electric motors drive the five main movement axes (dosage, injection, plasticizing and injection unit, machine movement and ejection). The control cabinets are on one hand necessary for accommodating the mold and on the other hand for raising the clamping forces. These forces prevent the mold from opening during the shaping process. Another one of their tasks is closing and opening the mold. The mold is responsible for the shaping of the plastics and releasing the warm air so that a molded part can be formed out of plastics melt. The plasticizing and injection unit is used to melt and supply the plastic. The plastic granules are added through a hopper into a screw that is surrounded by a heated cylinder. The movement of the screw and the thermal conduction of the cylinder plasticizes the granules into a melt, which is pushed into the antechamber of the screw. The melt is then pushed forward into the closed mold. The control unit is needed for adjusting and observing the setting parameters whereas the temperature control unit regulates the temperature of the mold at a pre-set temperature [1].

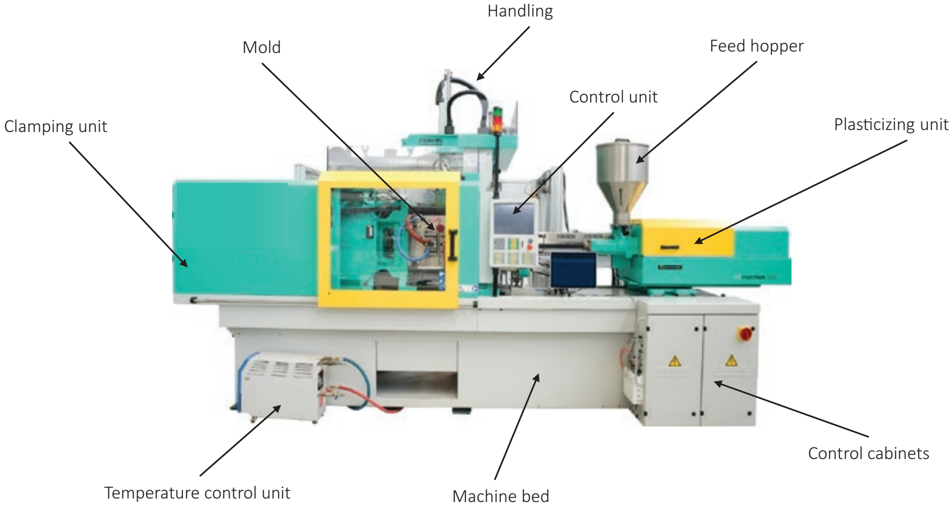


Figure 1.1 Set-up of an injection molding machine

Injection molding machines can be differentiated on the basis of their design and drive type. Thus, a distinction can be made between horizontal and vertical injection machines and fully electric, hydraulic and hybrid machines. The difference between a horizontal and vertical injection machine is the position of the mold or the molded parting line in the machine. The type of drive used determines whether the injection machine is fully electric, hybrid or hydraulic. The axes of fully electric machines are driven by an electric motor. Hydraulic machines are driven by hydraulics while hybrid machines have a mixture of hydraulics and electrically driven axes [2].

■ 1.2 The Injection Molding Cycle

A typical injection molding cycle runs in five steps. A new cycle commences when the mold closes. As soon as the mold is closed and the clamping forces are built up, the plasticizing and injection unit is pushed forward onto the mold. Then, the molten plastic (plastic melt) is injected into the cavity under a speed-controlled axial advance of the screw. The holding pressure phase commences when the mold is 95–98% filled and serves to regulate shrinkage of the molded part. In this phase, the holding pressure is regulated and the plastic melt is pushed further into the mold to balance the volume shrinkage caused by the temperature. The phase ends when the connection of the sprue to the molded part solidifies. This means that the sealing point has been reached. From this point on, the plasticizing and injection unit of the mold returns to its original position and the so-called residual cooling time begins. During this phase, the dosage process for the next cycle begins, which means that plastic melt again fills up the antechamber of the screw. As soon as the molded part reaches its ejection temperature, the mold opens and ejects the molded part. Alternatively, a handling system removes the molded part from the mold. Figure 1.2 shows the injection molding process.

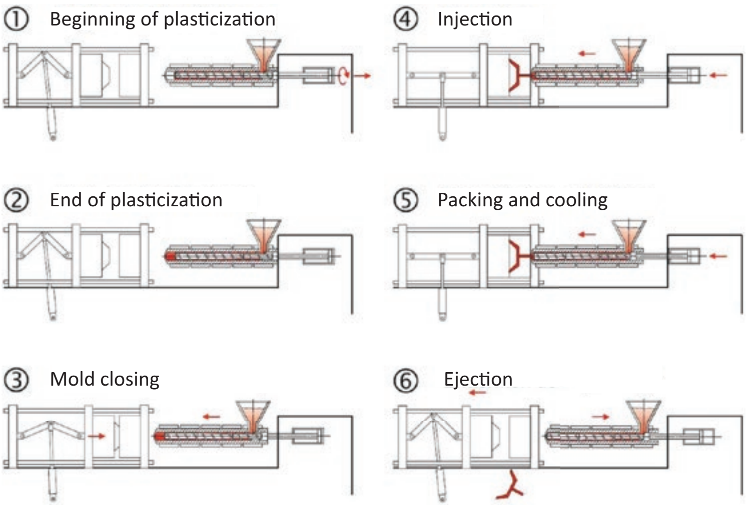


Figure 1.2 The injection molding cycle [1]

The cycle time consists of the time needed for opening and closing the mold, the deployment of handling and ejectors, as well as the injection time, the holding pressure time and the residual cooling time. It is essential for the financial planning to give consideration to the cycle time, as reducing it is one of the most important aims of the injection molding process. Based on the cycle time, the mold and number of cavities can be calculated for the required number of molded parts per year. Figure 1.3 shows a typical injection molding cycle [2, 1].

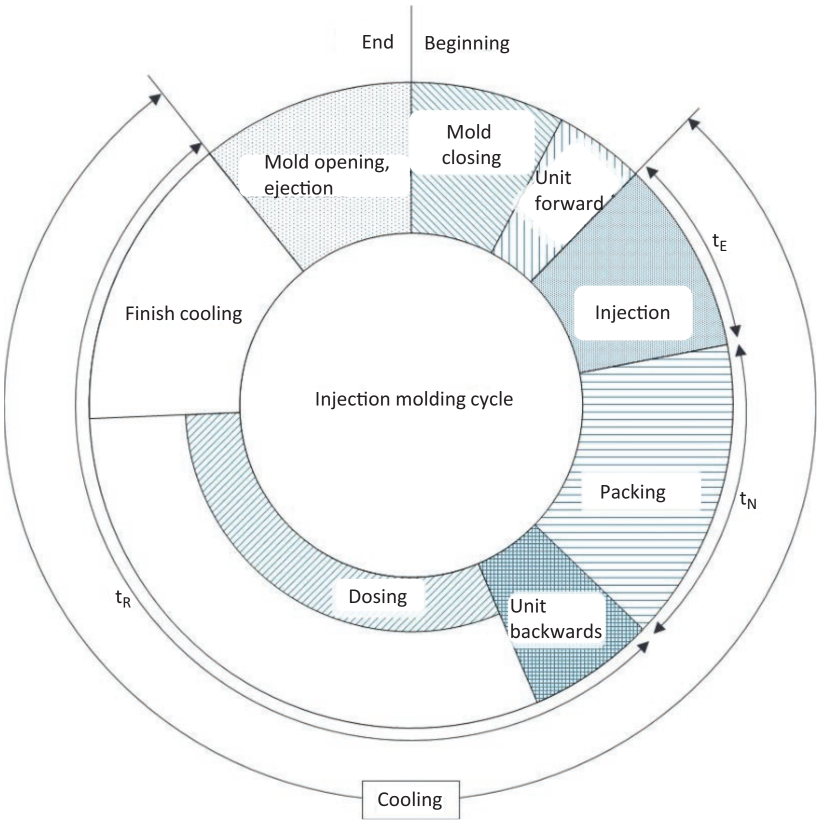


Figure 1.3 Injection molding cycle

The run time of the cycle can be calculated as follows (Equation 1.1):

$$t_z = t_E + t_N + t_{RK} + t_{WZopen} + t_{Ejector/Handling} + t_{WZclose} \tag{1.1}$$

Where:
t_Z = Cycle time
t_E = Injection time
t_N = Holding pressure time
t_{RK} = Residual cooling time
$t_{Ejector/Handling}$ = Ejection time
$t_{WZ\ open/close}$ = Mold opening and closing

As soon as the mold is closed, the injection phase begins. This phase also affects the cooling time. The reason for this is that, when the material enters the mold, the plastic melt touches the wall of the mold and that starts the cooling process. The holding pressure phase is needed to balance the volume shrinkage when the molded part solidifies. During this phase, a specified holding pressure pushes more plastic melt into the mold.

Immediately after the holding pressure phase is concluded, the residual cooling time before the molded part is ready to be ejected begins. During the residual cooling time, the granules for the next shot are being prepared, that is to say, plasticized. Once the molded part is stable enough to be ejected, the mold opens and the part can be pushed out of the mold by the ejectors. It is also possible to take out the part by means of a handling system. The time required for this needs to be taken into consideration when calculating the cycle time.

1.3 The Injection Mold

Generally, each injection mold consists of two mold halves (Figure 1.4): one is fixed (nozzle side) and the other is movable (ejector side). The molded part is formed by the cavity. The plastic melt streams through the sprue bush until reaches the cavity. The sprue bush is located on the fixed-nozzle side and, during the injection phase, it is in contact with the plasticizing and injection unit. In general, the cavity consists of a cavity plate, which usually is on the nozzle side, and a core that sits on the ejector side.

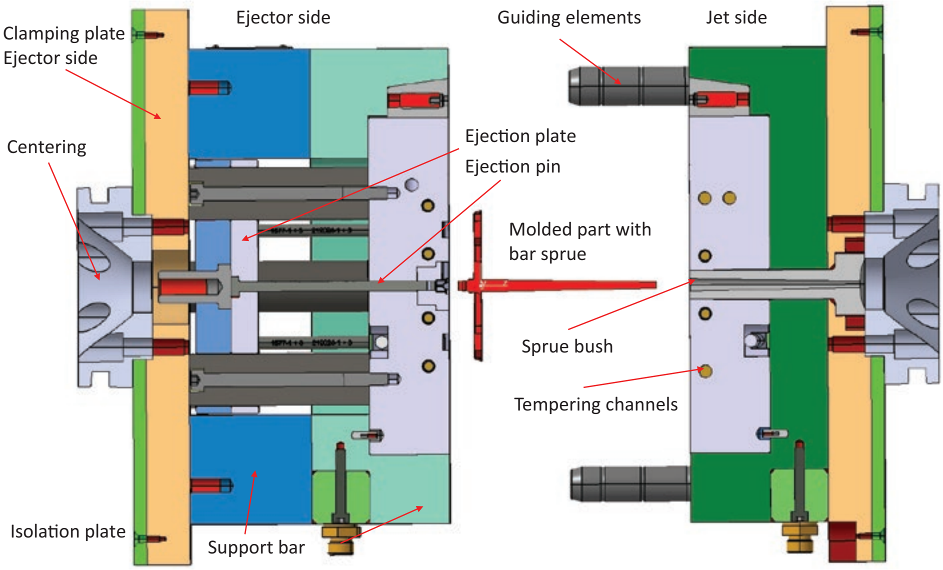


Figure 1.4 Structure of a two-plate injection mold (Source: PlastSolutions GmbH)

Cooling channels in both mold halves are necessary to dissipate the heat of the molded part. A cooling medium (usually water) flows through these cooling channels. With water and under pressure, cooling is possible up to 200 °C. For higher temperatures, oil is used in most cases to dissipate the heat. However, water has better heat transfer properties than oil. The goal is to achieve turbulent flow, as that allows for better heat transfer. The cavity plate as well as the core are located in the mold plate. After the molding process, when the molded part has reached the ejection temperature, the ejector pins remove the molded part from the core on the ejector side. During this process, an ejector bolt is pushed forward, usually by a hydraulic ejector. The ejector plate in which the ejector pins are located push the molded part out of the ejector side.

The supporting bars are behind the intermediate plate and in between the supporting bars is the ejector plate. For fixing the mold on the movable machine plate, a clamping plate is located at the end of the ejector side. The same applies to the nozzle side. Here, the clamping plate is located at the end of the fixed plate to attach this side to the fixed machine plate. If necessary, an additional isolation plate can be added to these to plates to reduce the heat transfer from the mold to the clamping plate. It is possible to install a centering ring on the nozzle side as well as the ejector side. This aids the installation of the mold inside the machine and the centering of the mold (center line to the plasticizing unit). Guide elements guide the mold and hold it together when not assembled.

One of the main tasks of the injection mold is to let the plastic melt stream from the plasticizing and injection unit through the sprue bush and finally into the cavity. In the cavity, the heat of the plastic melt needs to be reduced so that the molded part can be cooled down to ejection temperature. This means that the mold is tasked with giving the molded part its form and transferring the heat. After the ejection temperature is reached, the ejectors push out the molded part. In general, it is important that the mold is rigid enough to withstand the high pressure during the injection phase as well as deformation due to the high clamping force.

Injection molding simulation programs can interpret injection molds rheologically, thermally, and mechanical. This means that it is possible to simulate the flow of the plastic melt from the tip of the screw, to the sprue system to the end of the flow path. Finally, the injection molding simulation program is able to simulate the holding pressure and cooling phases. This allows the pressures, speed, shear rate and stress, temperatures and other measurements to be depicted during the injection, holding pressure and cooling phases. The injection molding simulation process and the cooling can be optimized and the shrinkage and warpage can be predicted. A prediction of the shrinkage and warpage enables early optimization of the injection mold and, if necessary, adjustment of the mold [3]. Statistical methods (Design of Experiments or DoE) support the user of the simulation program during the evaluation of the major parameters on the qualitative features and the process optimization [4].

Current simulation programs make it possible to simulate the starting process of an injection molding machine. As it takes some time for the thermal balance of the injection mold to be reached, simulation of the startup process is especially important. In addition to the deformation of individual mold elements, it is also possible to predict the offset of the core. A mechanical interpretation of the mold can also be executed by the simulation programs.

1.4 The Phases of Injection Molding – Forming the Molded Part

Generally, the molded part is produced in three phases:

- injection phase (dynamic phase)
- compression phase (quasi-static phase)
- holding pressure phase (quasi-static phase)

These three phases define the quality of the plastic molded part. Figure 1.5 shows the cavity pressure gradients and the three phases of molded part formation during injection molding.

All the main quality features, such as weight, dimensional stability, surface quality and so on are mostly formed during these three phases. Therefore, it is crucial to know the pressure gradients during these phases. Computer-aided simulation programs can be of service here as well.

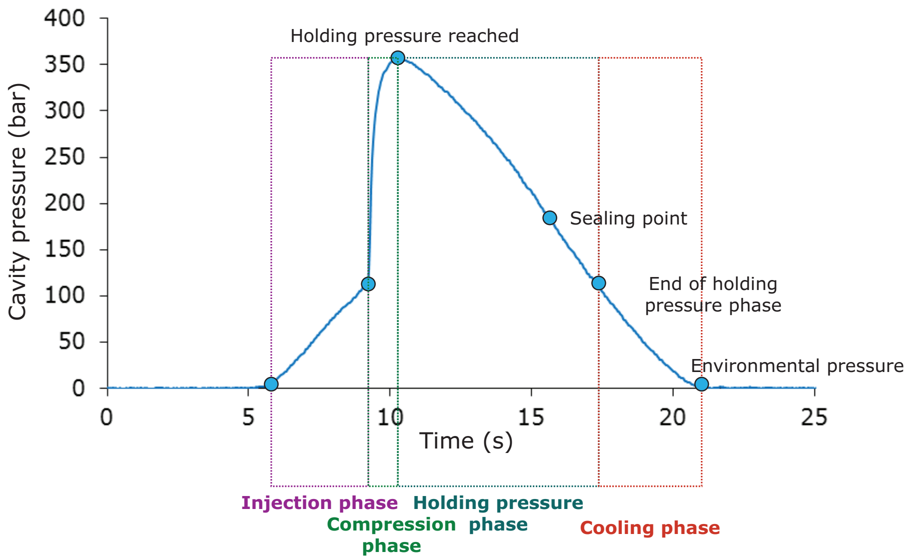


Figure 1.5 Cavity pressure curve during injection

These programs help with making statements about the formation of the molded part (pressure, flow rate, shear, shear stress, temperature, etc.). As the process of forming the molded part (Figure 1.5) is directly connected to the quality of the molded part, it is essential to be aware of possible weak spots or faults in the molded part in advance. A high level of expert knowledge is generally necessary since the correlations in this process are usually very complex.

1.4.1 Injection Phase

In the dynamic injection phase, the liquid plastic that has been prepared by the plasticizing and injection unit is injected into the closed mold by the axial advance of the screw. This process is usually speed-controlled. This means that the injection molding machine provides the injection pressure (up to 2600 bar) necessary for the filling of the mold. Therefore, it is crucial that the switching point, where the speed-controlled injection phase switches over to the pressure-controlled holding pressure phase, is precisely timed. The switching point has to occur somewhere between 95 and 98% of volumetric filling of the cavity. The injection phase is followed by the compression phase.

In the injection phase, the surface quality of the part is defined by fountain flow at the melt front and the wall adhesion of the plastic melt. Furthermore, the orientations, caused by the flow direction, of molecular chains and fillers, such as glass fiber, are formed. If the switching point takes place at the right time, the cavity pressure will look similar to Figure 1.6.

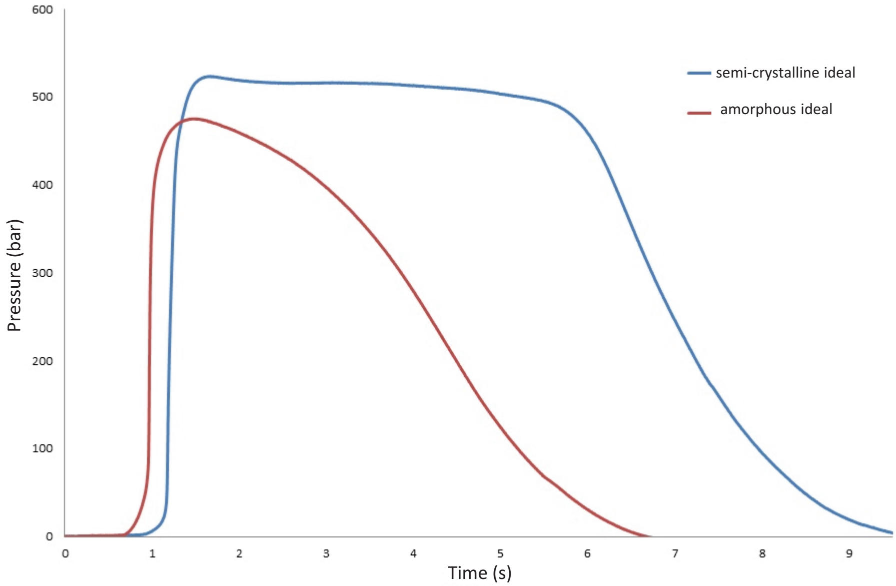


Figure 1.6 Cavity pressure at the right switching point during injection molding

Figure 1.6 shows the cavity pressure for an amorphous polymer (red) and a semi-crystalline polymer (blue). The curve for the amorphous polymer tends to be harmoniously round, due to the cooling behavior. On the other hand, the curve for the semi-crystalline polymer shows a strong holding pressure level at first. Next, the cavity pressure decreases rapidly. The standard switching point in simulation programs is usually set at 98–99% volumetric filling. This value does not necessarily have to be the correct point for the switching process. The same applies to the injection time or speed. The plausibility of these standard values should always be checked in a simulation.

1.4.2 Compression Phase

The quasi-static compression phase follows the injection phase. This phase heavily depends on the switching point. If the switch from injection to compression happens too early, the cavity pressure will collapse (Figure 1.7, left). Also, the volumetric residual filling of the cavity is pressure-controlled and this affects the quality of the molded part (surface, orientations, etc). Even if the switch has only been executed after 99% of volumetric filling, the injection molding machine will try to keep the injection speed in a constant balance, even if the cavity is already completely filled. This leads to high pressure peaks inside the mold (Figure 1.7, right).

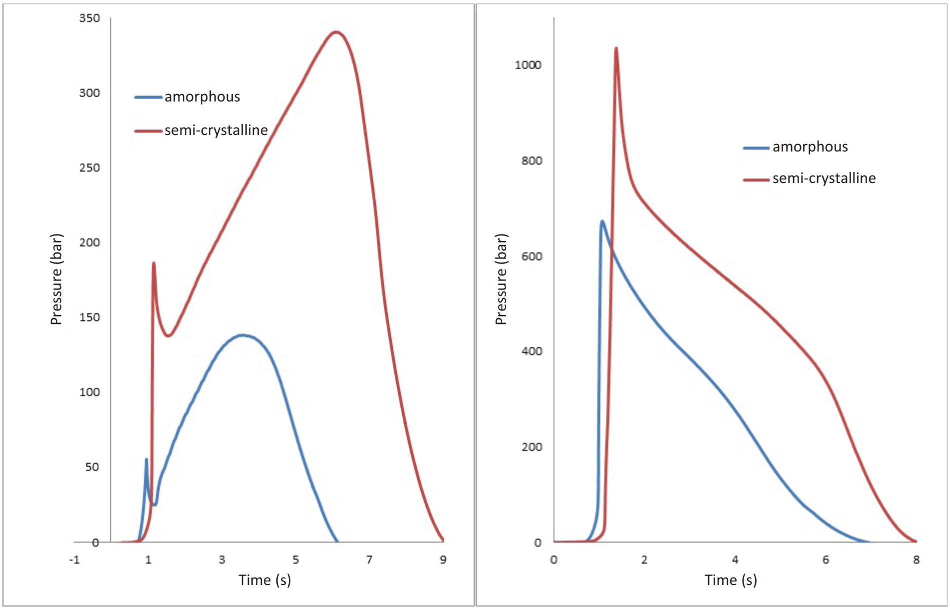


Figure 1.7 Cavity pressure at the wrong switching point during injection molding

In some circumstances, the clamping force exerted by the injection molding machine may not be sufficient if the switching point is set too late. As a result, the mold may open. This can cause overfilling and, directly related to this, the formation of burrs. During the cooling phase, the burr remains in the mold parting line. In this phase, the molded part peels away from the cavity surface. During this process, the clamping force of the machine puts pressure on the burr (web) that is situated in the molded parting line. As a result, damage to the mold may occur. Additionally, the high cavity pressure may cause the plastic melt to be pushed back into the plasticizing unit. This is possible when the cavity pressure is higher than the set holding pressure and when the machine or the mold has no needle gate nozzles.

1.4.3 Holding Pressure Phase

The quasi-static holding pressure phase is supposed to compensate (balance) for volume shrinkage of the molded part. The volume shrinkage, which is caused by the cooling of the molded part and the related relaxation processes of the molecules, is compensated by the further feeding of hot plastic melt under high pressure. The user of an injection machine usually chooses a percentage of the pressure (the melt pressure in the ante-chamber) as the starting point for the holding pressure. A starting point could be 30% of the injection pressure. This starting value can be increased until the desired part quality (dimensional accuracy, sink marks, voids, weight, etc.) is reached. The holding pressure should only be set as high as necessary, as a high holding pressure can impair part quality (residual stresses, orientations, etc.). Additionally, a high holding pressure puts strain on the machine and the mold, such that energy usage increases.

The simulation programs usually have a guide value, mostly listed in tables, for the level of the holding pressure. This value depends on the type of plastic and the part geometry. The simulation programs do not know the injection pressure, more specifically the melt pressure in the ante-chamber. Therefore, they need to use the values in the tables for their calculations. It is therefore necessary to check the values and compare them to practical experience. The same considerations also apply to the holding pressure and the cooling time.

The holding pressure time ends after the so-called sealing point has been reached. Usually, the sprue that has a thin wall freezes first and causes the on-going holding pressure to stop. The curve for the cavity pressure shows the sealing point of an amorphous polymer very clearly (Figure 1.7, red) at the turning point of the decreasing holding pressure.

The simulation programs use these curves not to calculate the sealing point, but to define a flow limit or no-flow temperature. This no-flow temperature is calculated from the pvT data and the specific heat capacity. The is the limit value which determines when the plastic freezes and stops melting.

1.4.4 Cooling Phase

The plastic part is cooled down to ejection temperature in the cooling phase. In the literature, a distinction is drawn between a medium and a maximum ejection temperature. While the cooling equation for the medium ejection temperature is more likely to be used for amorphous polymers, the equation for the maximum ejection temperature is preferably used for semi-crystalline polymers. A precise value is set in the simulation programs. The ejector temperature can be calculated from the curve of the 1 bar line in the pvT diagram. For amorphous polymers, the ejector temperature is 30 to 50 °C below the glass temperature (kink in the 1 bar line). For semi-crystalline polymers, a tangent is drawn to the curve of the transition area from fluid to solid. The point of intersection is the ejection temperature.

■ 1.5 Molded Part Faults

Several different faults can occur on the plastic molded part during the injection molding process as a result of a variety of influential parameters (settings, machine, mold, material, etc). As a wide range of aspects could be the cause of these faults, it is often difficult to identify them and in turn to solve the issue. Many faults or flaws can be identified in advance by the simulation. Consequently, appropriate measures can be applied in the simulation to rectify the issues. Therefore, executing the simulation early can help to reduce costs significantly and avoid time-consuming corrections on the mold later on.

1.5.1 Gloss Differences

During the injection molding process, differences in the gloss of the molded part surface can appear. There are two reasons for their occurrence: either more or less severely glossy spots are formed on the part surface despite a uniformly structured cavity, or the gloss of the whole surface area is too intense or weak. In general, the intensity of the gloss is impacted by molding of the surface cavity (Figure 1.8). It is possible to achieve a more matt surface when the molding has been done well. In addition, the surface of the mold must have some structuring, as these bumps result in a diffuse, non-targeted reflection of light. However, if the surface of the mold is polished, a well-executed molding process results in a more intense glossy surface of the molded part. The mold and melt temperature are critical to these processes, as well as the injection temperature and the holding pressure. The settings significantly influence the development of the frozen edge and therefore also the molding of the surface. Fundamentally, it needs to be noted that the surface quality of a plastic molded part is primarily defined by the flow of the melt in the injection phase.

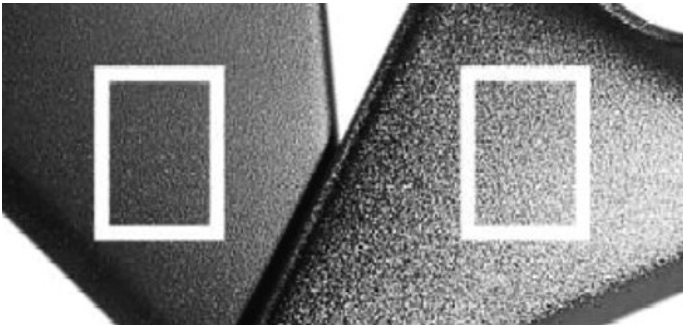


Figure 1.8
Gloss differences

Possible solutions:

- Optimize the injection speed (usually: increase the speed).
- Increase the mold temperature (the temperature limit has to be observed). However, a higher mold temperature extends the cooling time and the cycle time.
- Increase the melt temperature (the temperature limit has to be observed). The melt temperature also influences the length of the cooling time but not as much as the mold temperature.
- Increase the holding pressure and, if necessary, extend the holding pressure time.
- Delay the switching point to holding pressure until the fill level has reached 98% to achieve a process optimization.

1.5.2 Knit Lines

Knit lines (weld lines) cannot be avoided during injection molding (Figure 1.9). They can develop behind a breakthrough, through several flow paths, as a result of inserts, through differences in wall thickness, or due to several gating points. Knit lines develop when two melt flow fronts collide more or less head-on. Notches can appear on the surface of the plastic part near the knit lines. In most cases, differences in gloss and color are also visible. Knit lines can be notably spotted on transparent or dark plastic parts with a polished surface or plastics which contain effect pigments or glass fibers. If the melt meld lines meet under overly low pressures and temperatures, notches form on the surface in this area. The melt meld lines then do not properly weld together. Due to this reason, mechanical strength is most likely reduced under these circumstances. Another problem that can occur with knit lines is the orientation of the reinforcing fillers such as glass fibers. The smaller the angle between the melt flow fronts, the more extreme the distinctness of the knit line usually is.

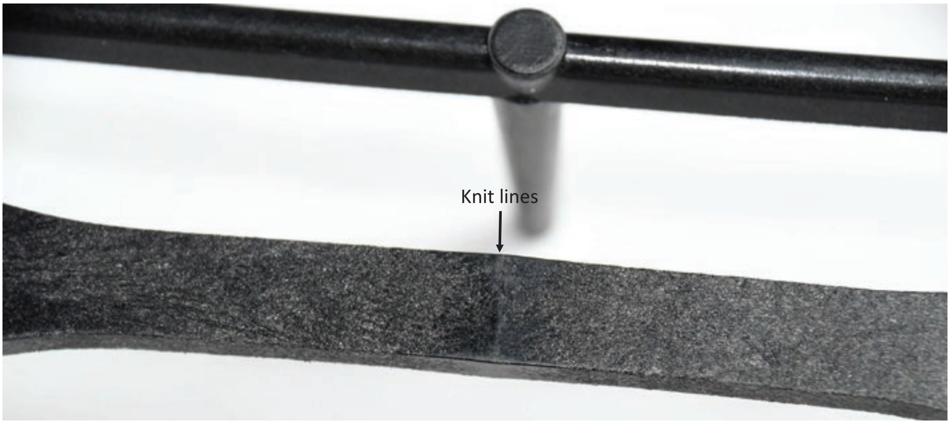


Figure 1.9 Knit lines

Possible solutions:

- Optimize the injection speed. If the venting is good enough, the injection speed should be increased.
- Use a filling study of the plastic molded part to check the venting. If the air cannot escape properly near the knit line, it will be compressed. This may result in the diesel effect. In this case, it is necessary to implement venting in this area. Venting can be achieved by the implementation of more ejectors or special inserts, such as disk packs, as well as a splitting of the mold insert into diverse small parts.
- Increase the mold temperature (the temperature limit has to be observed). However, a higher mold temperature extends the cooling time and the cycle time. It is also possible to regulate the temperature of local inserts in the mold (variothermal dynamic temperature; see Chapter 10). This way, it is possible to heat the knit lines in the injection phase locally. As a result, the quality might increase or the knit lines may even become invisible.
- Increase the melt temperature (the temperature limit and elongation of the cycle time have to be considered).
- Delay the switching point to holding pressure until the fill level has reached approximately 98%; this could lead to process optimization.
- Increase the holding pressure to improve the welding of the two melt flow fronts.

- (Reinforcing) fillers and pigments usually impair the quality of the knit lines.
- Move the gating point to influence the position of the knit lines due to the wall thickness. If necessary, the wall thickness can be adjusted. However, retroactively this is only possible by modifying the mold. Therefore, the effectiveness of this measure should be checked in advance with a filling simulation. The simulation programs show the knit lines in color as a function of the angle of incidence. Consequently, adjustments to the geometry of the plastic part, such as the gating point, are easy to make.

1.5.3 Deformation

Deformation can occur when overly strong or irregular forces impact the ejection of the plastic part (Figure 1.10). Usually, this can be seen in the material, which expands, tears, or even breaks due to the high ejection forces. In most cases, in contrast to warping, deformation occurs near the area of the ejector or on undercuts that are difficult to demold. Moreover, deformation can cause deep mold marks or scratches in the direction of demolding.

Figure 1.10 Deformation

Possible solutions:

- Check the draft angles. After they have been evaluated, the drafts should be increased if necessary.
- Check the position and the number of ejectors.
- To avoid deformation, adjust the holding pressure.
- Add a lubricant or anti-adhesion coating to the cavity.
- Reduce the temperature of the mold core.
- Optimize the core venting. This can be achieved by installing more venting, special inserts, or ejectors, etc., through which the air can flow between the molded part and the core.

1.5.4 Warping

Differences in volume shrinkage in different areas of the molded part can cause deviations from the planned shape. These deviations are called warping (Figure 1.11). It is visible through warps, wave-like surfaces, torsion or deviations in angles. The difference between holding pressure forces (shrinkage compensation) and orientations cause the differences in shrinkages. Another cause of warping could be internal stress.

Figure 1.11
Warping

Possible solutions:

- Cool the molded part evenly. This means that the temperature of the mold needs to be checked.
- Ensure sufficient compensation of volume shrinkage in the holding pressure phase. To achieve this, choose a holding pressure that is sufficiently strong and long.
- To avoid warping, increase the injection speed.
- Uniform filling process of the mold is advisable.
- Choosing a material that flows more easily can be beneficial.
- Another feature should be considered when choosing the material: low shrinkage values. Semi-crystalline and unfilled plastics tend to shrink more than plastics with fillers and amorphous polymers.
- Several sprues improve the holding pressure forces and therefore reduce warping.
- Uniformly distributing the melt along the flow path could lead to less warping.
- Avoid sharp corners.

- Reinforce areas where warping primarily occurs (ribs).
- Check whether the fiber orientation can be changed.
- Prevention of warping should already be considered when manufacturing the mold by ensuring that the molded part with warping corresponds with the reference part.
- Reduce differences in wall thickness and unusual accumulations of melt.
- Warping can be minimized if convex areas or edge beads in which the deformation has no impact can be formed. Simulation programs can predict warping and suggest preliminary corrections in the mold, Cadmould with Unwarp, for example. However, these suggestions need to be critically examined before implementation. For this reason, Cadmould issues a warning (see Chapter 10).

1.5.5 Diesel Effect (Burners)

If the air cannot escape properly during the injection phase from the mold, black discoloration may appear near the knit lines, where the melt flows collide, or in specific areas, such as the ribs or at the end of the flow path. In this case, the air gets compressed during the filling process, so that the diesel effect occurs (Figure 1.12). In isolated cases, it can also prevent the molded part from filling completely. This could also result in a layer forming on the surface of the mold or even thermal damage. The compressed air gets heated so much that burned spots or burners can appear in this area.

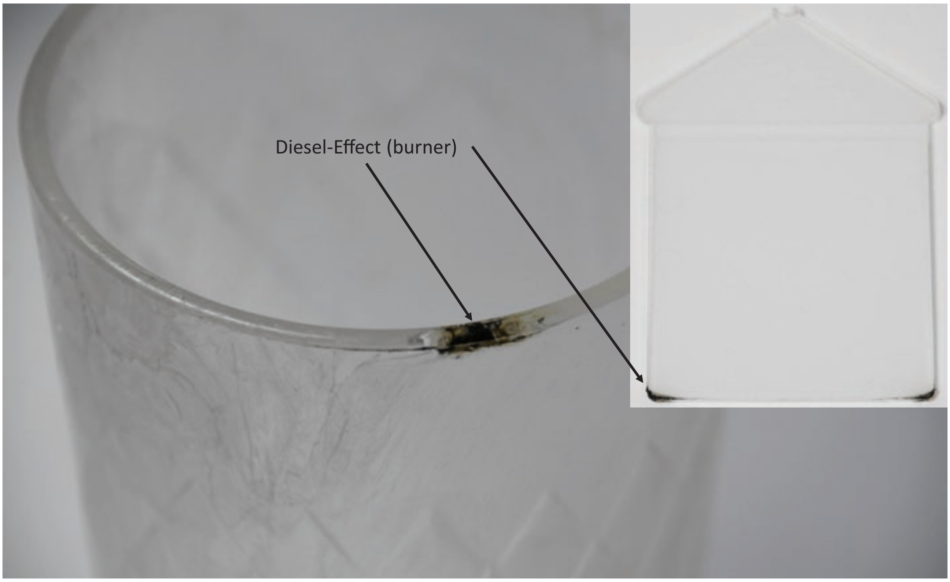


Figure 1.12 Diesel effect (burner)

Possible solutions:

- To avoid the diesel effect, install venting in the mold. Moreover, special inserts for venting, joints and ejectors could be installed to help the air escape.
- Check the venting channels for dirt.
- Optimize the filling process of the molded part accordingly so that air can escape through the joints.
- Lower the melt temperature and the mold temperature to achieve a positive effect.
- Reduce the screw advance speed to decrease the injection speed. Alternatively, it may be sufficient to run a graduated injection profile.
- If the burn spots occur mostly near the parting line, reduce the clamping force of the injection machine.
- All simulation programs indicate the areas where venting should be positioned in the mold. These suggestions can be useful.

1.5.6 Mold Coating (Deposits)

If temperatures are too high during production or if venting is inadequate, a coating can form on the injection mold. The formation of this coating is the result of plastic by-products, or additives, reacting with each other (Figure 1.13). These by-products consist of cracked polymers or other products of decomposition, such as flame retardants.

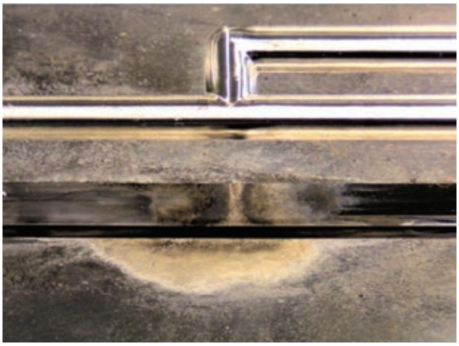


Figure 1.13
Mold coating

Possible solutions:

- Reduce the screw advance speed to decrease the injection speed. Alternatively, it may be possible to run a graduated injection profile, which reduces the speed only in the final phase of filling.

- Compare the gating with the design standards for the sprue size. Make appropriate corrections if execution is poor.
- Compare the melt temperature at the point of exit from the machine nozzle or the hot runner with the recommended values using a probe thermometer. If there are any deviations, adjust the temperature accordingly.
- Check the residence time of the melt in the plasticizing unit and in the hot runner.
- Reduce shearing in the plasticizing unit for a sensitive melt by reducing the screw speed and the back pressure.
- Review the effectiveness and the position of the venting channels and install additional venting channels, if necessary.

1.5.7 Black Specks

Not only thermal damage, which decomposes polymers, but also dirt can cause black specks. These take the form of plate-like or point-like deposits on the surface and the inside of the molded part which are especially visible in transparent materials (Figure 1.14).

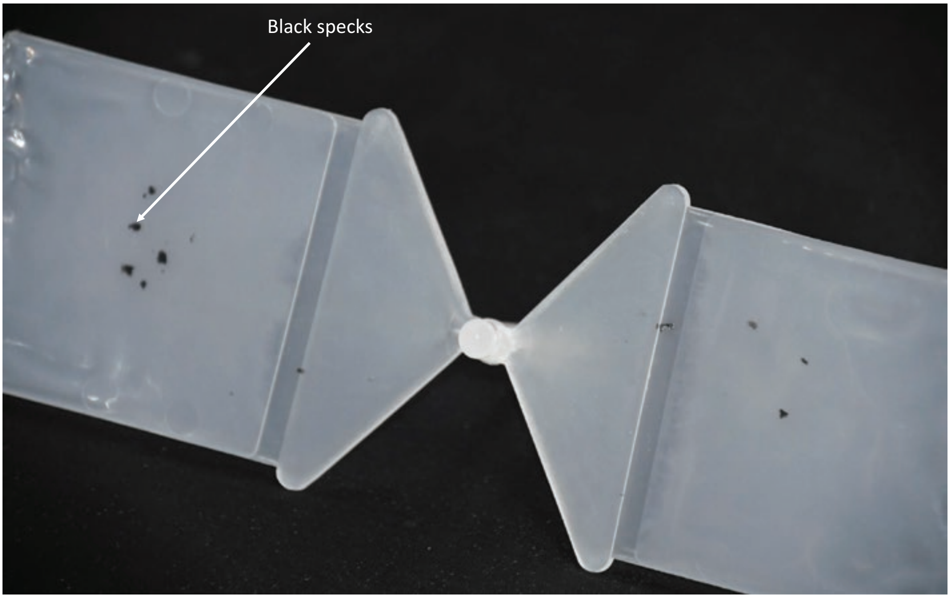


Figure 1.14 Black specks

Possible solutions:

- Check the metering stroke of the injection machine. Optimally, the metering stroke s_D is double the screw diameter D. In general, the following applies: $D < s_D < 3D$. If the metering stroke s_D is less than the screw diameter, the residence time of the melt in the plasticizing unit is too long. Check the residence time of the melt in the hot runner, where present.
- Reduce the melt temperature.
- Reduce the back pressure and/or the speed of the screw.
- If the material needs to be changed, the plasticizing and injection unit as well as the hot runner, if necessary, need to be cleaned thoroughly.
- Avoid dead spots by checking the flow of the melt in the areas of the cylinder and the mold.
- Analyze whether the masterbatch or powdered pigment can be used in combination with the type of plastic that is to be processed.
- Check the functionality of the coating of the mold and/or the plasticizing unit (where present).
- If a material-conveying system is present, check it for abrasion, leaks and dirt deposits.

1.5.8 Ejector Marks

Ejectors can also cause optical damage in the form of differences in gloss on the molded part. Possible faults can show up as differences in gloss, stress whitening (white discolorations) or elevations and indents (Figure 1.15). In some cases, scratches or grooves can occur in the demolding direction. This can be caused by production errors in the mold, such as insufficient stiffness and overly small drafts. Moreover, if the draft angle of the molded part is too small, it may be more difficult to demold a part and this can cause marks. In addition, if the mold is overloaded due to unfavorable production conditions, the molded part can end up being jammed, which also makes demolding more complicated. This applies to high-volume shrinkage as well, which can cause severe shrinkage of molded part onto the core.

Figure 1.15
Ejector marks

Possible solutions:

- Possibly extend the change-over time and/or the post-cooling/residual cooling time.
- Adjust the holding pressure; this generally means reducing it.
- Lower the processing temperatures.
- Treat the cavity with a lubricant or non-stick coating.
- If the ejectors do not match the surface of the mold, make appropriate modifications. However, this only applies in the case of marks and not stress whitening.
- Enlarge the draft angles, if necessary.
- Enlarge any existing ejectors and/or consider further ejectors.
- Strengthen the mold.

1.5.9 Burn Streaks

Severe thermal damage of the plastic melt can lead to burn streaks on the surface of the plastic part (Figure 1.16). The reason is that bubbles develop in the injection molding phase as gaseous fractions of the plastic are released. The bubbles shear at the wall of the cavity, which results in irregular light brown and silver colored, but also dark discoloration. This discoloration can be visible on the surface of the plastic part or in the gating area.

In most cases, the type of discoloration already indicates the root cause of the damage. For example, a long machine standstill while the cylinder heating is still on can often lead to oxidation or disintegration. The severe thermal damage resulting from this becomes visible as light brown to dark brown streaks. In contrast, silver streaks are often the result of strong friction. They are often limited locally to narrow cross-sections of the flow channel or the areas around small nozzles.

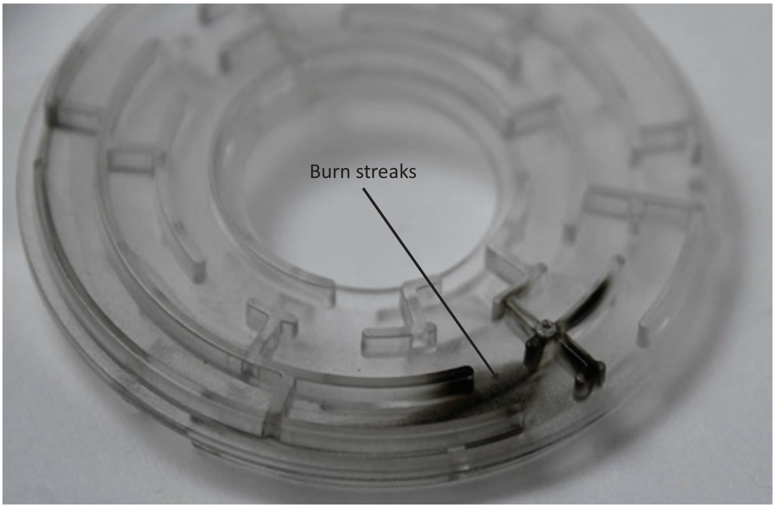


Figure 1.16 Burn streaks

Possible solutions:

- First, check whether the machine was at a standstill before the streaks occurred.
- The plasticizing unit should be checked for its size, that is to say, the residence time. If necessary, swap it out. Twice the screw diameter D is the optimal metering stroke s_D. Generally, $D < s_D < 3D$. If the metering stroke s_D is less than the screw diameter, the residence time of the plasticizing unit is definitely too long.
- Use a probe thermometer to measure the temperature of the melt in the machine nozzle. The temperature should be reduced if it is not in the recommended processing area.
- Additionally, check the temperature of the melt exiting the hot runner. If it exceeds the recommended range, adjust the temperature.
- Reduce the back pressure. Then, evaluate the effects that this change has on the streaks.
- Vary the screw speed to evaluate its impact on the streaks.

- Reduce the injection speed.
- Check the gate geometry and adapt if necessary.
- Avoid sharp transitions and narrow flow cross-section in the cavity, where possible.
- Optimize the flow cross-section of the machine nozzle and/or the hot runner.
- If there is a hot runner, the melt should not remain in it for too long. The residence time can be decreased by reducing the cycle time as a whole. Apart form that, the hot runner needs to be re-dimensioned.

1.5.10 Sink Marks

Sink marks can be another type of surface damage (Figure 1.17). Especially, areas of melt accumulation, such as in the foot of ribs, can be affected. If the design guidelines cannot be maintained when constructing the molded part, local melt accumulations can lead to increased shrinkage in this area. This causes the surface coating to be pulled in. If the surface does not yield, voids can appear instead of sink marks in the molded part.

Often sink marks only develop after the part has been demolded from the mold, as the already cooled edge layers get reheated by the heat coming from the inside of the molded part and they can therefore be deformed again.

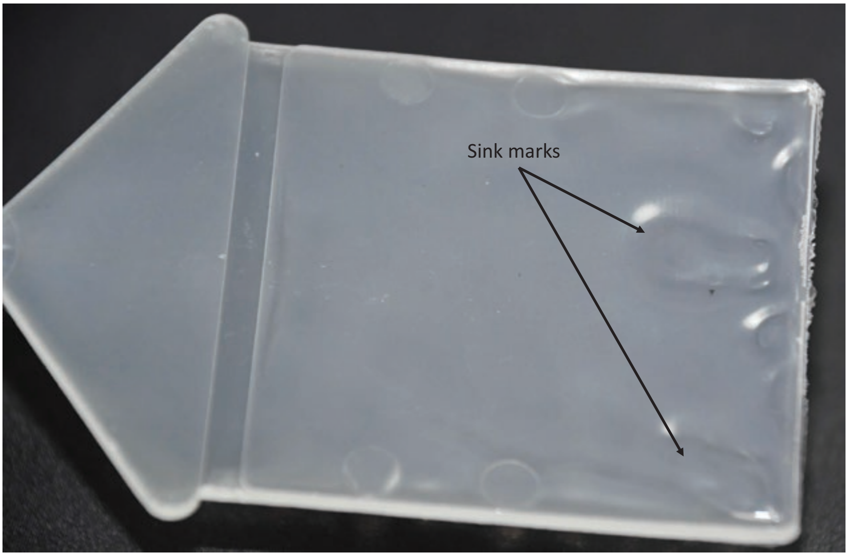


Figure 1.17 Sink marks

Possible solutions:

- To reduce the sink marks, lower the mold and/or the melt temperature.
- Set the holding pressure to a higher setting or, if necessary, extend it. This concerns especially the phase immediately after volumetric filling.
- Alternatively, consider extending the residence time.
- Extend the holding pressure by one or two seconds longer than the sealing time. To calculate the sealing time, the duration is increased in small steps, starting with a short holding pressure time. The weight of the molded part is measured at the corresponding holding pressure time. When the weight remains constant, the sealing point has been reached and the holding pressure can no longer act.
- Reduce the injection speed.
- If the gating point is in an area with a thin wall, consider moving the gate. Doing a simulation in advance could help.
- Enlarge the gate.
- Another measure is to design thinner ribs.
- The screw stroke during injection as well as during application of holding pressure should not sink to zero. Therefore, check the residual melt cushion to ensure that the size or weight of the molded part is not smaller than 3–5 mm.
- Furthermore, check the functionality of the non-return valve. A fluctuating residual melt cushion can indicate a defective non-return valve.
- Cool the molded part once more after the demolding.
- Sink marks can be visualized by the simulation program. The user can therefore examine appropriate measures in advance and adapt them accordingly.

1.5.11 Record Grooves (Cold Meld Lines)

Parallel or concentrated grooves are called record grooves or cold meld lines. They form diagonally to the flow direction and leave marks on the surface of the molded part (Figure 1.18). If the flow rate is too slow, even for a short amount of time, it can lead to cooling of the convex melt front and will result in this effect. The areas of the melt front which are too close to the wall of the mold, cool down too much. Therefore, it cannot be guaranteed that the melt is pressed completely against the mold wall despite the continuously rising pressure and progressive filling. This causes the development of grooves on the surface of the whole melt front.

Another cause can be a temporary melt standstill in front of a thin rib or close to another narrow location. Additionally, changing to holding pressure too early can cause the record grooves effect.

Figure 1.18
Record grooves

Possible solutions:

- Increase the injection speed to reduce the effect.
- Increase the melt temperature, in the hot runner as well, if necessary. The recommended temperature limit should not be exceeded.
- In addition, raise the temperature of the mold, but only to the recommended temperature limit. Increasing the process temperatures also always extends the cooling time.
- The change to holding pressure can be delayed until filling is approximately 98% complete. This may optimize the process.
- Consider changing the location of the gate or the thickness of the wall.

1.5.12 Incomplete Filling of the Mold

If the mold does not fill completely, the molded part may not be fully formed in areas with thin walls or in areas that are distant from the gating point (Figure 1.19). A low dosage (residual melt cushion), not enough pressure, back flow in the return pressure valve or an excessively strong resistance during filling can lead to this effect. The filling resistance consists of the length of the flow path, the wall thickness, and the viscosity.

Figure 1.19 Incomplete filling of the tool

Possible solutions:

- First, check the residual melt cushion and the metering stroke.
- Choosing a more readily flowing material can help to fill the mold better.
- Another option is to increase the melt and/or mold temperature. It is, however, advantageous to first increase the melt temperature, because increasing the relative mold temperature means that the filling pressure cannot be reduced as much. Moreover, increasing the mold temperature can extend the cooling time much more than were the melt temperature to be increased.
- Avoid excessive pressure loss in the gating point and in the cavity.
- Switching from the injection pressure to the holding pressure later can have a positive effect.
- Alternatively, optimize the injection speed. Usually, it needs to be increased.
- Optimize ventilation at the end of the flow path. In the area of the air pocket, install a vent.
- Balance out the filling process.
- Moreover, adjusting the filling pattern by installing flow brakes or aids can lead to an improvement.

- Consider moving the gate. To facilitate, this simulation technology can be used.
- In general, simulation programs can be a useful mold for avoiding incomplete filling.

1.5.13 Color Streaks

If pigments of a powdered pigment, masterbatch or liquid pigment are unevenly homogenized in the plastic matrix, color differences or streaks can occur on the surface of the molded part (Figure 1.20). These streaks occur close to and far away from the gating point and over large areas near the meld lines. They may even occur behind sharp-edged spots. In these areas, pigments accumulate and form color agglomerates, which are responsible for the color differences. The agglomerates near the surface are visible as streaks, whereas the pigment agglomerates in the inner areas of the molded part wall cause patchy color differences.

There are several causes for this molded part flaw: pigments that are not a match for the plasticizing unit, due to an incorrect *L/D* ratio, or the wrong processing parameters, for example if the melt temperature is not high enough. On the other hand, colored or carrier parts which are incompatible with the plastic to be colored might trigger irregular colored patches or streaks.

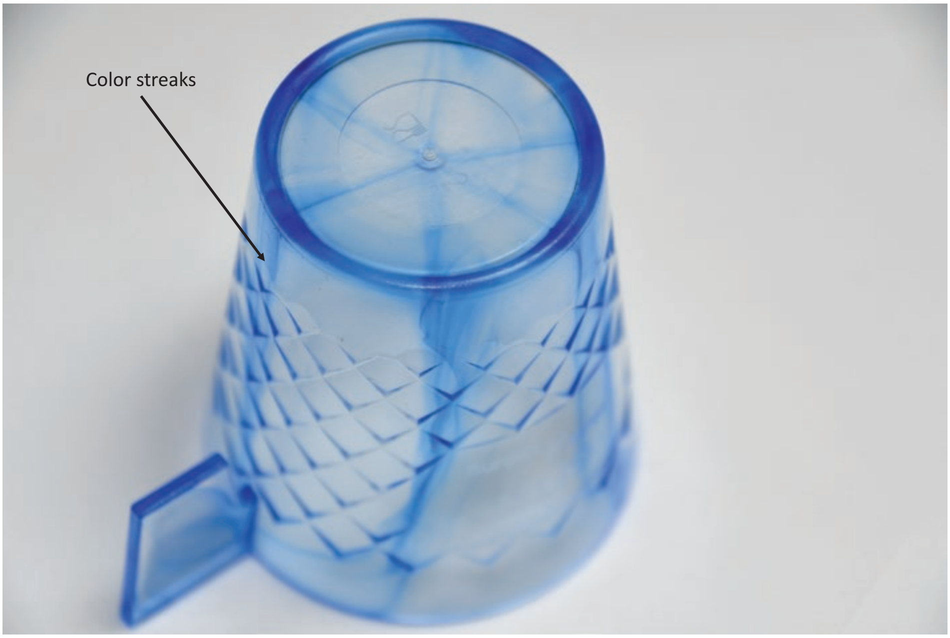


Figure 1.20 Color streaks

Possible solutions:

- First, check the masterbatch data sheet, which is provided by the supplier, for compatibility of the masterbatch with the chosen plastic material. Simultaneously, compare the settings for the masterbatch concentration with those given in the supplier information.
- Moreover, checked whether the temperature of the hot runner is compatible with the masterbatch and whether it falls within the processing range. Heat sensors and the regulation of the hot runner system should be checked for this.
- Increase the injection speed to decrease color streaks.
- Increase the back pressure to help avoid pigment accumulations.
- After increasing or reducing the speed of the screw, analyze the effect on the color streaks.
- To reduce the formation of the color streaks, check whether the melt temperature at the point of entry into the mold is high enough.
- Alternatively, move the gate or adjust the wall thickness.
- Check the plasticizing unit for its size, which means the residence time, or, if necessary, swap it out. The optimal metering stroke s_D is twice the screw diameter D. In general, $D < s_D < 3D$. If the metering stroke is less than the screw diameter, the residence time of the melt in the plasticizing unit is in any event too long. The residence time should be compared against that given in the supplier information. Moreover, the screw specifications (L/D ratio, mix and/or shear component necessary, etc.) should be checked against requirements.

1.5.14 Glass Fiber Streaks

Irregular and rough patches on the plastic mold surface are called glass fiber streaks (Figure 1.21). In some cases, they also occur in regular intervals at the front of the melt. Depending on the angle of the light, they might have a more metallic shimmer or be rather dull. Glass fiber streaks can form especially at turnarounds, meld lines or breakthroughs, because flow processes near ribs and very thick walls and the processing parameters (mold and melt temperature, injection speed) heavily influence this formation.

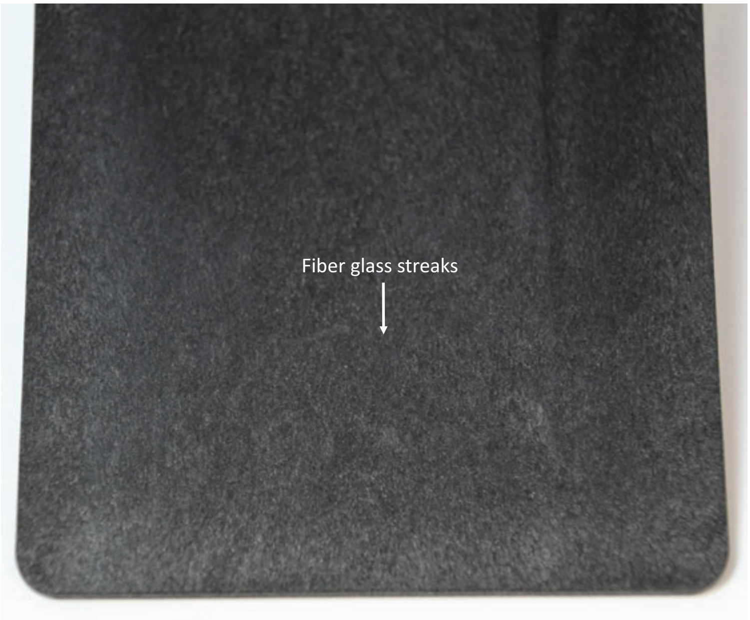


Figure 1.21 Glass fiber streaks

Possible solutions:

- Reduce glass fiber streaks significantly by optimizing the homogeneity of the melt. To achieve homogeneity, increase the screw speed and/or the back pressure.
- Increase the mold and/or the melt temperature (if necessary, also in the hot runner). However, keep in mind the recommended temperature limit and the cooling time.
- Increase the screw advance time, if possible, by using a step-by-step injection profile.
- Check whether the gate can be moved.

1.5.15 Moisture Streaks

Another type of streak can occur on the surface of the plastic part: moisture streaks (Figure 1.22). These can form if the residual moisture in the granules or in or on the mold is too high. The latter occurs when moisture is released through leaks in the temperature-control system or through water of condensation. The streaks stand out due to their elongated, parabolic form. However, it should be noted that their tip always points in the direction of the flow path. The moisture causes visible marks on the surface of the plastic part, as steam bubbles form in the injection phase, which then pop on the melt front. Due to the fountain flow, they get pushed to the surface. Consequently, the bursting water and gas bubbles cause elongated streaks.

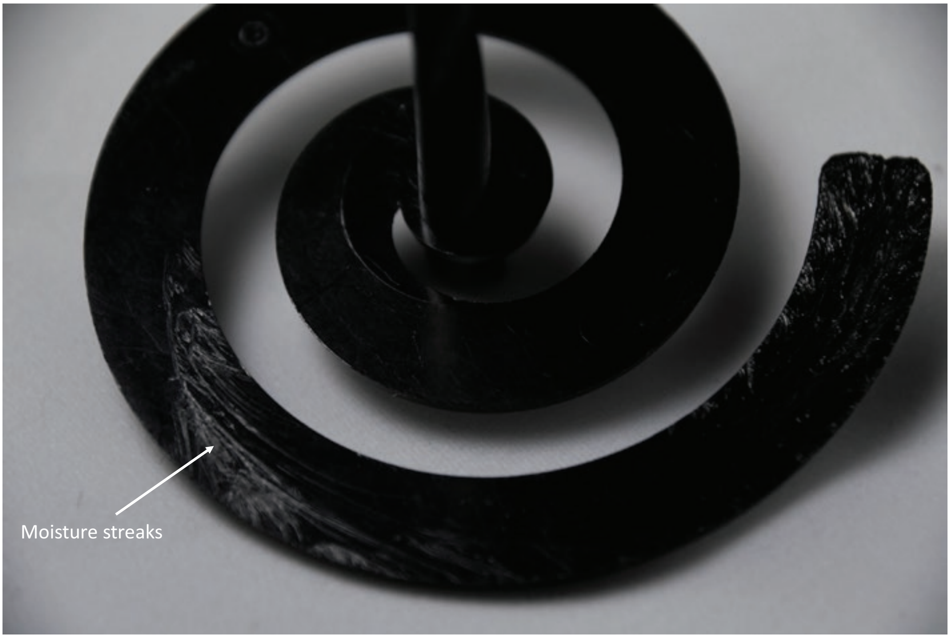


Figure 1.22 Moisture streaks

Possible solutions:

- First, check the granules packaging for external damage.
- Especially in the case of hygroscopic plastics, which tend to absorb water (PA, PET, PC), the residual moisture of the granules must be measured. It is essential to adhere to the maximum allowed level of residual moisture. Otherwise, it may not only cause streaks on the surface but also lead to hydrolytic degradation of polymer chains.
- Reduce the amount of granules in the material hopper.
- Check whether the storage conditions of the granules match the suggestions provided by the raw materials supplier. Furthermore, check the settings for pre-drying.
- Examine the inside of the mold as well. The temperature-control system should not have any leaks and no water of condensation should form on the mold surface.

1.5.16 Delamination

If there is not enough of a connection between the surface layers of a plastic part, delamination can occur. This means that the surface of the plastic part splinters or that it fans out (Figure 1.23). This can also occur when the already cooled edge layer is moved by the shear stress induced by the flow process and therefore peels off. Depending on the type of plastic, this has different impacts on the plastic part. If it is a semi-crystalline polymer, then the layers have different crystal structures. Amorphous thermoplastics, on the other hand, tend to become segregated due to a reaction in the additive/melt/pigment mix.

Figure 1.23 Delamination

Possible solutions:

- Compare the current settings with the settings of the last process that was completed successfully.
- Using the masterbatch data sheet provided by the supplier, check whether the masterbatch is compatible with the chosen plastic material.
- Reduce the injection speed. Simultaneously, increase the manufacturing temperature.
- Moreover, always thoroughly clean the plasticizing unit before a change of materials.
- Avoid sharp turnarounds, especially near the gate, as they can cause the material to shear heavily. Additionally, avoid high wall shear stress (see Chapter 16).

1.5.17 Tiger Stripes

A pulsating melt flow, which is mostly the result of a blend (thermoplastic multiphase system), can lead to markings diagonally to the flow direction. These markings become visible on the plastic part surface. As the periodic shadows resemble the pattern on the coat of a tiger, they are also called tiger stripes (Figure 1.24). Generally, the elastic features of plastic materials are responsible for this phenomenon. Plastic materials have more or less elastic features. The elastic part functions as a spring and causes the pulsation. This phenomenon is particularly likely to occur when pressure is relieved on the plastic melt and the melt enters the mold in this state (see Chapter 9).

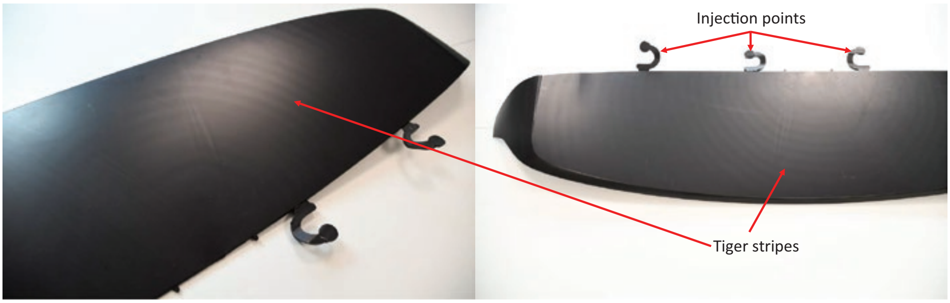


Figure 1.24 Tiger stripes

Possible solutions:

- Enlarge the wall thickness of the plastic part and increase the injection diameter to avoid the occurrence of tiger stripes.
- Moreover, increase the mold and/or melt temperature to help avoid this defect. However, adjust the melt temperature first, as this is often more effective and does not seriously extend the cycle time.
- Furthermore, choose a different material that has better or different flow abilities. In general, a material with less elastic qualities (memory module) can reduce tiger stripes.
- In cascade injection, avoid pressure pulses. Simulation technology can offer a lot of support here.
- Tiger stripes can to a certain extent be visualized by simulation programs. Usually, pressure fluctuations are responsible for this effect, which is why they can be simulated through special hot runner technology. This can aid the finding of possible solutions.

1.5.18 Formation of Stress Cracks, Micro Cracks

If stress inside the material is lower than the tensile stress at break, cracks can occur on the inside and outside of the plastic part. Prior to the occurrence of so-called stress cracks, crazes form in the material (Figure 1.25). Crazes are crack-like, extended areas, which are connected through fibrils, i.e., highly stretched strands of molecules. They tear apart, among other things, when local stress and other tensile stress impact them from the outside. Aggressive media can also be responsible for the cracks. An increase in the notch effect or the spreading and swelling pressure influences the cracks. In addition, the stress within the plastic part, whose strength is impacted by the manufacturing process, also affects the cracks. Due to the cooling processes of the melt in the cold mold, it is not possible to avoid all stress in the plastic part. The aforementioned media, however, can detect the stress caused by the manufacturing process.

Figure 1.25
Formation of stress cracks, micro cracks

Possible solutions:

- Increase the mold temperature without exceeding the temperature limit. It can be particularly advantageous to cool down both walls of the plastic part evenly via the cooling circuit temperature.
- Additionally, reduce the holding pressure.
- Stiffening the mold design can help to avoid micro cracks.
- Generally, avoid melt accumulations and, if necessary, reduce them. For example, the wall thickness of a rib in the foot of the rib can be minimized.
- Stress can also be visualized by the simulation program.

1.5.19 Jetting

With this type of plastic part damage, a strand of plastic melt forms, starting at the gate. It then develops in a snake-like pattern in the cavity and becomes visible in the plastic part (Figure 1.26). Normally during melt flow, the melt is in contact with the wall at the transition point to the large diameter. If, however, the flow rate is too high, the melt will hit the empty space in form of a jet. The melt folds at the point where the jet randomly and briefly strikes the wall of the mold. As soon as the surface of the strand cools, it cannot connect properly with the following melt and so visible damage occurs and the strength of the plastic melt decreases drastically. Jetting cannot be reproduced as it changes shape and spread with each shot. Therefore, it makes no sense to predict jetting with a simulation program. The program always assumes wall adhesion.

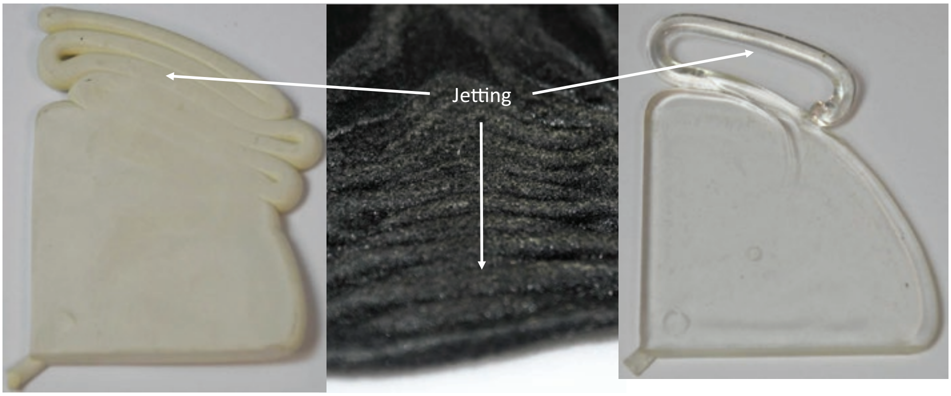


Figure 1.26 Jetting

Possible solutions:

- To avoid jetting, reduce the injection speed.
- Concerning the gate, either enlarge the gate or move it to an area of smaller flow diameter.
- Jetting can be prevented by rounding the transition between plastic part and gate.
- Furthermore, the plastic melt should be faced with an obstacle immediately after the gate.
- As previously established, a jetting cannot be sensibly simulated. On one hand, simulation programs assume wall adhesion, an assumption which contradicts the occurrence of jetting. On the other, jetting cannot be reproduced and can therefore not be simulated.

1.5.20 Voids

After the plastic part has cooled down, microcellular or bubble-like cavities, which are also sometimes called vacuoles, can form inside the plastic part. If one drills into the plastic part in colored water and these do not fill with water, gas bubbles have formed inside the plastic part. However, if it is a truly empty space and water can get in, voids have formed (Figure 1.27). Unlike gas bubbles, voids often occur near the plastic core, which is situated in the middle of the wall or next to melt accumulations. Like sink holes, voids form in areas of high volume shrinkage where melt compression due to the holding pressure is insufficient. Unlike sink holes, voids can also appear when stiff edges are frozen and therefore not flexible anymore. The contraction forces acting as the plastic melt cools inside the part cannot be compensated anymore. As a result, empty spaces form in these areas.

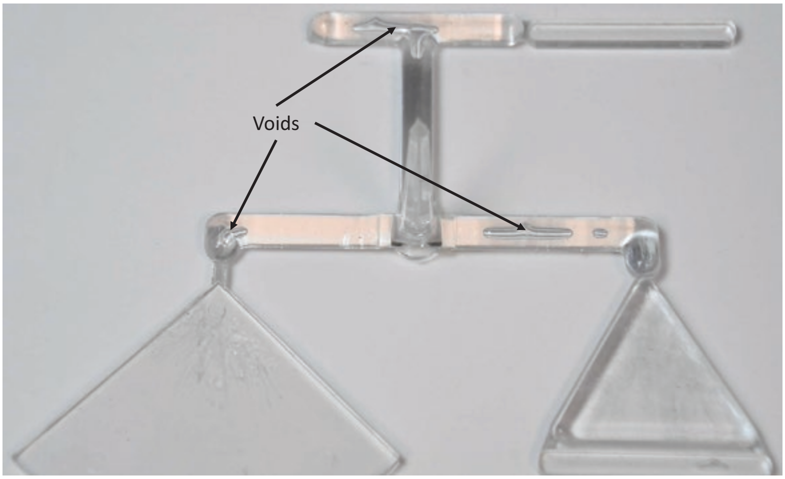


Figure 1.27 Voids

Possible solutions:

- Enlarge the gate. Similarly, move the gate if the gating point is near an area with a small wall thickness. Simulation can be very useful here.
- When designing the ribs, etc., observe the design guidelines.
- The holding pressure time should be one or two seconds longer than the sealing time. To detect the sealing time, a brief holding pressure time is increased in small steps. The weight of the part is measured at each holding pressure time. If the weight remains constant, it means that the sealing point has been reached and that the holding pressure is no longer acting.
- Another solution would be to increase the holding pressure.

Index

D

E

F

Q

R

S

T