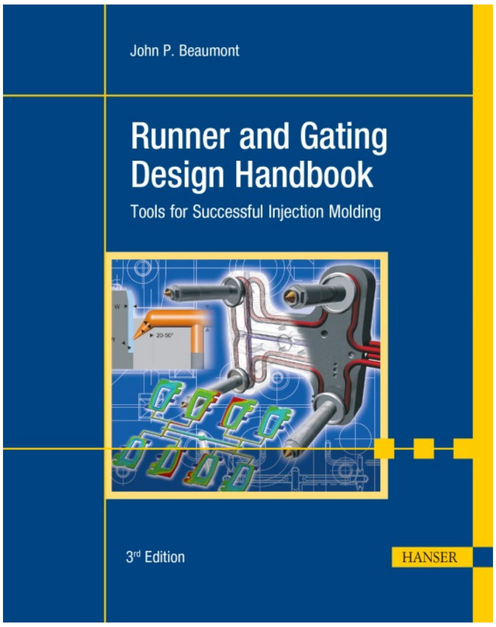
HANSER



Sample Pages

Runner and Gating Design Handbook

John P. Beaumont

ISBN (Book): 978-1-56990-590-6
ISBN (E-Book): 978-1-56990-591-3

For further information and order see

www.hanserpublications.com (in the Americas)

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

Preface

Quality management methods, such as *Design for Six Sigma*, stress the critical review of fundamentals in order to identify and eliminate potential problems before they take their toll on the manufacturing process. In developing a mold design to produce an injection molded plastic part, one of the most fundamental and influential components is its melt delivery system. It also turns out that the melt delivery, or runner, system is probably the most underappreciated and misunderstood component of the injection mold. This makes it a prime candidate for critical review, particularly for the conscientious molder striving to improve his/her bottom line.

The melt delivery system begins with the injection molding machine's nozzle and continues into the mold, progressing through the sprue, runner, and gate. Though the melt may only experience these flow channels for a fraction of a second, their effects are dramatic and result in the most extreme conditions experienced by the plastic melt in any phase of nearly any plastics processing method. Shear rates in gates commonly exceed 100,000 s^{-1} and localized melt temperature in high shear laminates can spike at as much as 200 °C, at rates that can exceed 1000 °C/s. Due to the extremity of these conditions, the actual effect of these conditions on the melt is not well understood. Most material characterization methods do not even come close to measuring melt conditions under these extremes. Viscosity vs. shear rate data are generally developed at a maximum of 10,000 s^{-1}, DSC data at less than 32 °C/min, and PVT data at less than 3 °C/min. As a result of the limitations of material characterization methods as well as solution modeling and meshing issues, today's injection molding and fluid flow simulation programs are still struggling to accurately predict the extreme non-homogeneous asymmetric melt conditions developed in a branching runner. The challenge of dealing with these conditions has generally been underestimated.

The influences of these extreme melt conditions developed in the runner are just beginning to be understood. One of the most significant is the realization that the combination of laminar flow and high perimeter shear in a runner results in extreme non-homogenous melt conditions across a runner. Not only can a 200 °C variation in melt temperature exist but, as a result of the non-Newtonian characteristics of the melt, the viscosity may easily vary 100-fold from the zero shear conditions in the center of a flow channel to the extreme shear conditions around the perimeter. This creates significantly asymmetric melt conditions when the melt branches in a runner or part-forming cavity. The conditions developed in the runner continue into the part, corrupting the expected filling pattern and influencing how the part is packed, its mechanical properties, shrinkage, and warpage. These are all factors that are hardly known by most in the molding industry and their dramatic effects are rarely fully appreciated. The influence can be particularly acute in two-stage injection processes such as gas assist, structural foam, MuCell®, and co-injection.

As stated earlier, the melt delivery system consists of the molding machine's nozzle, sprue, runner, and gate. Each of these components, or regions, can have a significant influence on both the process and the molded part. Process effects include the ability to fill and pack the part, the injection fill rate, the clamp tonnage, and the cycle time. Effects on the part include size, weight, mechanical properties, and variations in these characteristics between parts formed in different cavities within a multi-cavity mold.

Despite the significant influence that the melt delivery system has on the molding process, its various components are generally poorly designed relative to the time, effort, and cost put into the other components/regions of a mold and molding machine. This book bridges the critical gap left by other publications dealing with injection molding, which generally touch only briefly on the design of the melt delivery system and its relationship to successful injection molding. In particular, the lack of information on cold runners needed to be addressed. Though a fair amount of published data on hot runners are available, these data are generally heavily influenced by the bias of companies that sell these systems. There are over 50 companies offering hot runner systems and components commercially, while there is no company at all offering cold runner systems. As a result, one can imagine the lackluster image of cold runners, as there is no company commercially promoting them.

Evidence of the lack of understanding of runners includes the fact that the significant effects of shear-induced flow imbalances in runners were not documented, or clearly understood, until 1997 when I published the first journal article on this phenomenon. For the first time, it became obvious that the industry standard "naturally balanced" runners were creating significant imbalances. Melt filling imbalances, developed from shear-induced melt variations, were found to be the norm in most of the industry standard geometrically balanced runner designs being used. This phenomenon was being overlooked by the entire molding industry for both cold and hot runner molds. In addition, the industry's leading state-of-the-art mold-filling simulation programs had been developed without the realization of the shear-induced imbalance. As a result, these programs did not predict the imbalance and left the analyst with a false impression that these runners provided uniform melt, filling, and packing conditions. The problem still exists today and should be considered when using analysis programs.

Of particular interest is the evolution of the runner from a basic necessity required to connect the injection unit and the mold's cavity to its emergence as a significant process tool. Newer melt rotation technologies, such as MeltFlipper® and iMARC™, have introduced the concept of 3D injection molding.

This book takes an independent view of both hot and cold runners, trying not to make a judgment as to which is best for a given application. Rather, it addresses some of the critical design issues unique and common to both. The early chapters lay a foundation for designing runners by establishing an understanding of the rheological characteristics of plastic melt and how the influence of runner design and gating positions can affect the molded part. Chapter 4 provides important strategies for runner designs and gating position, which are critical to the successful molding of a plastic part. Chapter 5 provides an overview of the melt delivery system, followed by Chapter 6 and 7, which teach the development and solutions to shear-inducted imbalances. These three chapters (5, 6, and 7) address issues which are common to both cold and hot runners, blending basic geometrical channel issues with melt rheology.

Chapter 8 focuses on cold runner designs including specific guidelines for runner and a wide variety of gate designs. Chapters 9 through 13 provide a close look at the design of hot runner systems and their unique capabilities and challenges. Chapter 14 provides a summary on the process of designing and selecting a runner system. Finally, the book concludes with an extensive troubleshooting chapter with contributions from John Bozzelli and David Hoffman.

This 3rd edition of Runner and Gating Design Handbook includes numerous updates and new instructional figures that are scattered throughout each of the 15 chapters. Chapters 6 and 7 include additional information and examples to aid in the understanding of critical shear induced melt variations that are developed in the runners of all injection molds. Autodesk Moldflow analyses and related discussions were added to help further understand the complexities of this phenomenon. Chapters 9 through 12 have expanded on all aspects of hot runners, including the design of manifolds, nozzles, gate tip designs, valve gated nozzles, and valve gate actuation. A new Chapter 15.3, “Injection Molding Process Development”, written by Dave Hoffman of the American Injection Molding Institute (AIM Institute), was added.

This book is intended to provide the reader with a better understanding of the critical role the runner plays in successful injection molding. It is hoped that this understanding should go a long way toward reducing mold commissioning times, improving product realization, increasing productivity, improving customer satisfaction, and achieving quality goals such as Six Sigma.

Contents

1 Overview of Runners, Gates, and Gate Positioning

In many cases, the mold design dictates the gating position, although ideally, the optimum gate position should be determined based on part requirements and afterwards the mold design selected to provide for the desired gate position. Available gating positions, and gate designs, are significantly influenced by whether the runner travels along the primary parting plane of the mold (the parting plane where the part forming cavity is defined) or whether it *does not* travel along this plane.

This chapter provides only a brief introduction and orientation of basic runner types and their influence on gate design and gating location. More detail on each of these subjects is presented later in the book.

1.1 Primary Parting Plane Runners

In the dominant runner type used in the industry the runner and part forming cavities are located along the same primary parting plane. Primary parting planes, often referred to as the parting lines, are where the mold opens and closes to allow ejection of the molded part and/or of the runner. The primary parting plane is the one where the molded part is formed and ejected. The *primary parting plane runner* is used in *two plate cold runner molds*. A cold runner mold is defined as a mold in which the plastic material in the runner is cooled and ejected from the mold during each mold cycle. Molten plastic material is injected through the runner, the gate, and then into the part-forming cavity. This molten plastic is then cooled by the mold, and when sufficiently solidified, the mold opens and the runner, gate, and part are ejected along the same primary parting plane. Figure 1.1 illustrates the position of the runner within the mold and its ejection from the primary parting plane. Notice that the part and runner are formed and ejected along the same parting plane.

After the molded part and runner are ejected, the mold again closes, creating a flow channel (runner path) between the injection molding machine nozzle to the part forming cavity. As the primary parting plane runner is located along the same parting plane as the part forming cavity, gating into the part is limited to its perimeter, or very near its perimeter. Sub gates, such as the tunnel, cashew, and jump gates, allow gating to be positioned within a short distance from the actual perimeter of the part (for gate designs see Section 8.4).

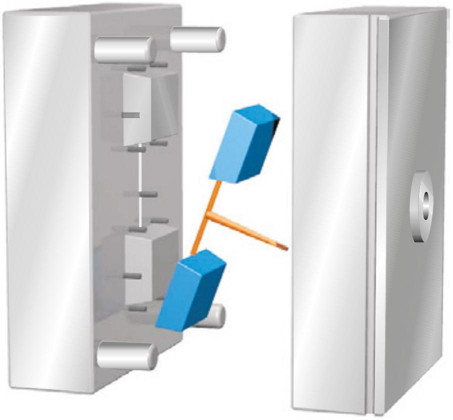


Figure 1.1 2-plate mold open and ejecting parts and runner

1.2 Sub Runners

A second runner type does not travel along the primary parting plane of the mold. This *sub-runner* generally travels parallel to the primary parting plane, but not along it. The sub-runner can be used in either a cold runner or a hot runner mold.

1.2.1 Cold Sub Runners

In a cold runner mold, the sub-runner travels along a second parting plane other than the primary parting plane where the part is formed. The two parting planes are normally parallel to each other and are separated, and partially defined, by at least one mold plate. The sub-runner and part forming cavities are connected by an extension of the sub-runner referred to as a *secondary sprue*. The bridging secondary sprue passes though the at least one separating mold plate and connects to the part-forming cavity through a small gate opening. The secondary sprues are normally parallel to the opening direction of the mold and perpendicular to the sub-runner (see Figure 1.2).

During molding, after the plastic melt in the runner and part forming cavity solidify, the mold will open along the two parting planes. The part is ejected from the opened primary parting plane and the runner (which includes the secondary sprue and gate) is ejected from the opened second parting plane as seen in Figure 1.3.

This type of mold is commonly referred to as a *three-plate cold runner mold*. The terms two-plate and three-plate cold runner molds refer to the minimum number of mold plates required to form and to allow removal of both the part and the solidified runner. With the two-plate cold runner mold, the part and runner are formed and removed between at least a first and second mold plate. With the three-plate cold runner mold, the part is formed and removed between at

3 Filling and Packing Effects on Material and Molded Part

The flow of thermoplastics through an injection mold and its relationship to the molded part is quite complex. This chapter focuses on the development of melt conditions within a part-forming cavity and their relationship to the molded part. This will help the reader establish an optimum gating and molding strategy.

3.1 Process Effects on Material Flow Characteristics

In Chapter 2, the basic behavior of thermoplastic materials was discussed and the relationships between a thermoplastic's viscosity, temperature, and shear rate were explained in detail. The initial viscosity of the melt entering a mold is determined by the melt temperature, as delivered from the molding machine, and the injection rate. High melt temperatures and high injection rates result in low viscosities for the plastic melt. This combination of high temperature and flow rate can result in lower fill pressures; however, pressure can begin to increase at extreme fast or slow fill rates. High melt temperatures are normally limited by potential degradation and longer mold cooling times. It is often desirable to perform a predictive mold filling analysis, such as with Autodesk Moldflow®, to determine the optimum balance of melt temperature, processing conditions (primarily injection rate), and runner diameter that will produce a quality product for a given part design. On the shop floor, use of molding techniques such as *Scientific Molding* [1] is commonly practiced to determine a target fill time for an existing mold. More recent methods for targeting an optimized injection molding process have been developed and are explained in Chapter 15 [2–4].

3.1.1 Melt Thermal Balance – Conductive Heat Loss vs. Shear Heating

The actual temperature of a melt in a mold is extremely complex. It not only varies along the length of a channel but can vary significantly across the channel. It is interesting to note that despite all of the scientific and technical advancements that have occurred since the introduction of injection molding, including putting a man on the moon and replacing the human heart over 50 years ago, we still cannot accurately measure the temperature of a melt in the mold. In recent years the best method to determine melt temperature is to calculate it using mold filling simulation programs. However, recognize that as we cannot measure the melt temperature, we cannot confirm the accuracy of the program's calculations.

During injection, a hot thermoplastic is forced into a relatively cold mold. As the melt travels through cold portions of the mold, heat is continually being drawn from the plastic material. Plastic directly adjacent the cold mold walls will freeze almost immediately. The thickness of the frozen layer is dependent on the balance between heat lost to the mold through conduction and heat gained from shear. If the injection rate into a mold for a thermoplastic material is too slow, the thickness of the frozen layer builds up to a point where material can no longer be fed into the cavity and a short-shot is created.

A short-shot is the extreme outcome when the injection rate is not adequate to keep the thermoplastic melt temperature elevated enough for molding. At faster fill rates, frictional heating can overcome the heat lost through conduction and allow the material to remain molten during filling of the entire cavity. Figure 3.1 shows the result of a series of mold filling analyses of a simple rectangular plaque at three different fill rates. The plaque is 50 mm wide by 150 mm long and 2 mm thick. It is edge-gated as indicated (along the bottom edges of the figures) and molded with an ABS and a melt temperature of 255 °C. Note the change in melt temperature and frozen layer variations in each of the figures dependent on flow rate. At the fastest flow rate, it can be seen that the melt temperature at the end of fill is actually 10 °C higher than the injection temperature.

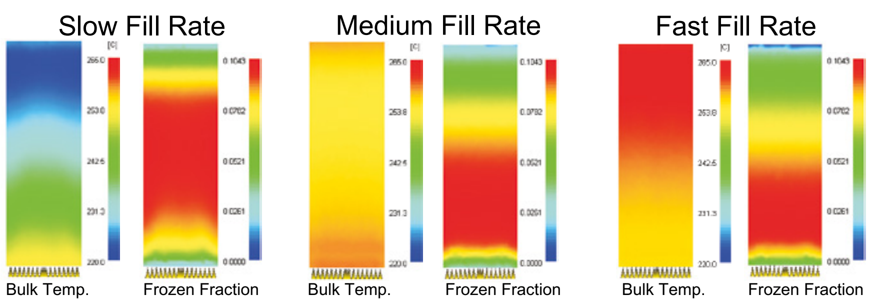


Figure 3.1 Effects of injection rate on bulk melt temperature and frozen material fraction as predicted by Moldflow's MPI

Control of frictional heating during mold filling can sometimes be difficult to achieve. With most parts, the geometry does not allow for the flow velocity of the melt to be constant without profiling the injection. Varying flow front velocities will result in a variation in the development of the frozen layer. A common example is the center gating of a disk-shaped part. At a constant injection rate from the injection molding machine, the flow front speed near the gate will be relatively high, but continually decreases as the melt progresses into the expanding cavity (see Figure 3.3). This will cause a high amount of shear heating near the gate, but as the melt front progresses, it slows down and will begin to lose more heat to the mold than it is gaining from possible shear heat. This effect can be minimized by utilizing an injection profile with an initial slower fill rate and then gradually increasing the injection rate. However, most molding is performed without the use of profiles.

Variations in wall thickness within a part can create significant variations in flow rate and the resultant thermal balance. Thin regions will create a resistance to the flow front and cause the melt to *hesitate* as it fills other thicker regions. The hesitating melt will quickly lose heat and potentially freeze off. This is discussed in more detail in Section 4.2.10.

A newer method *(Therma-flo™)* for mapping the injection molding characteristics of plastic materials can evaluate the effect of wall thickness, flow rate, melt temperature, mold temperature and length of flow [4]. A feature of this method includes the ability to determine a cavity fill rate at which the melt temperature from gate region to end of fill is uniform. This considers the thermal balance within the melt as a result of heat gain from shear versus the heat loss to the mold by condition. The red line in Figure 3.2 shows the change in the bulk flow front temperature of a PBT (Sabic Valox 420SEO) in a 2 mm (0.08 inch) thick channel after flowing 75 mm (3 inch). In this case it is shown that at a melt front flow velocity of 2 inch/sec, the melt temperature drops nearly 20 °F as it flows 3 inches, and increases by nearly 20 °F at a melt flow velocity of 25 in/sec. In this case, a thermal balance occurs at an in-cavity flow velocity of 4.1 inch/sec (10.4 cm/sec).

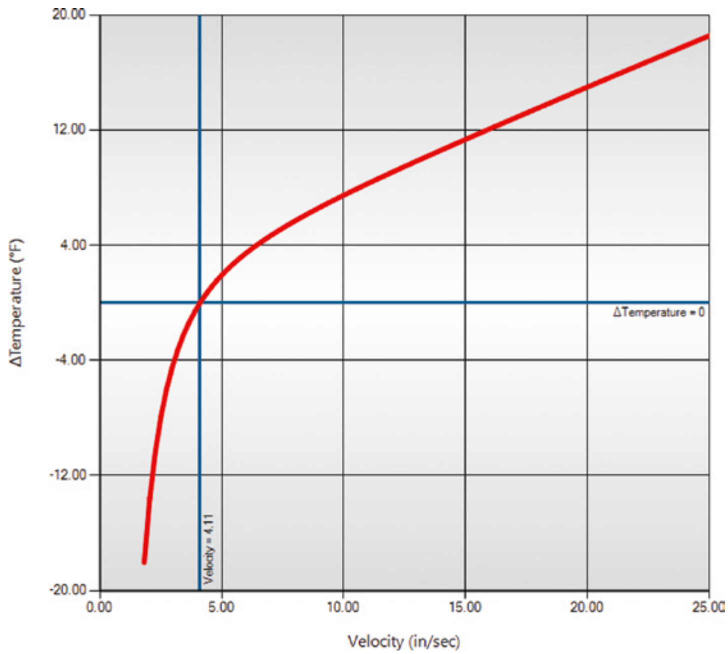


Figure 3.2 Thermal balance for a PBT in a 2.0 mm thick mold channel is shown to be occurring at a flow velocity of 4.11 in/sec (10.4 cm/sec)

There are many factors that contribute to the development of the frozen layer thickness in a molded part. The primary factors are:

- The thermoplastic's thermal properties (thermal conductivity, specific heat, and no-flow temperature, or transition temperature);
- The melt and mold temperature;
- The mold material's thermal properties;
- The local flow rate; and
- The residence time of the melt.

Figure 3.5 illustrates the distribution of frozen layer thicknesses that might occur between the gate and end of flow within a part having a diverging flow channel width such as a center gated disk. The frozen layer near the gate can be very thin because of the high shear rates and the constant supply of molten thermoplastic through the region of the part nearest the gate. The frozen layer is at its maximum thickness between the gate region and the flow front, and then again becomes relatively thin at the flow front due to the short time that the melt has been in contact with the cold cavity wall.

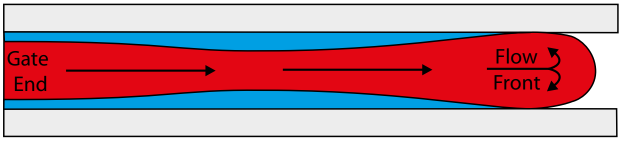


Figure 3.5 Development of frozen layer along the length of a polymer

Figure 3.6 is a summary plot from the Therma-flo™ moldometer showing the behavior of a polycarbonate (Covestro Makrolon 6455) [4]. Here pressure vs. flow front velocity at multiple wall thicknesses is shown. The results allow one to observe the contrasting impact of non-Newtonian shear thinning and the thermal exchange between melt and mold (including frozen layer development) vs. injection rate on mold filling pressures. Pressure (*y*-axis) is normalized by expressing it as pressure per length of flow (psi/inch). Velocity (*x*-axis) is the directly measured flow front velocity (inch/sec) of the melt in the monitoring channel. Note the pressure's reaction to flow velocity for each of the thicknesses shown (top to bottom curves represent cavity wall thicknesses of 0.06", 0.080", 0.100", and 0.140", respectively). Note that as flow velocity increases (left to right on the curve), pressure initially decreases as the melt benefits from non-Newtonian shear thinning, frictional heating, and reduction of frozen layer. As flow velocity continues to increase, there is a diminishing benefit of the non-Newtonian shear thinning and frictional heating. At some point the fundamental influence of the increasing melt flow rate of a pressure driven flow, and related flow velocity, becomes dominate and we see the pressure rise. The velocity at which the pressure is at a minimum is dependent on wall thickness and can be seen in this graph.

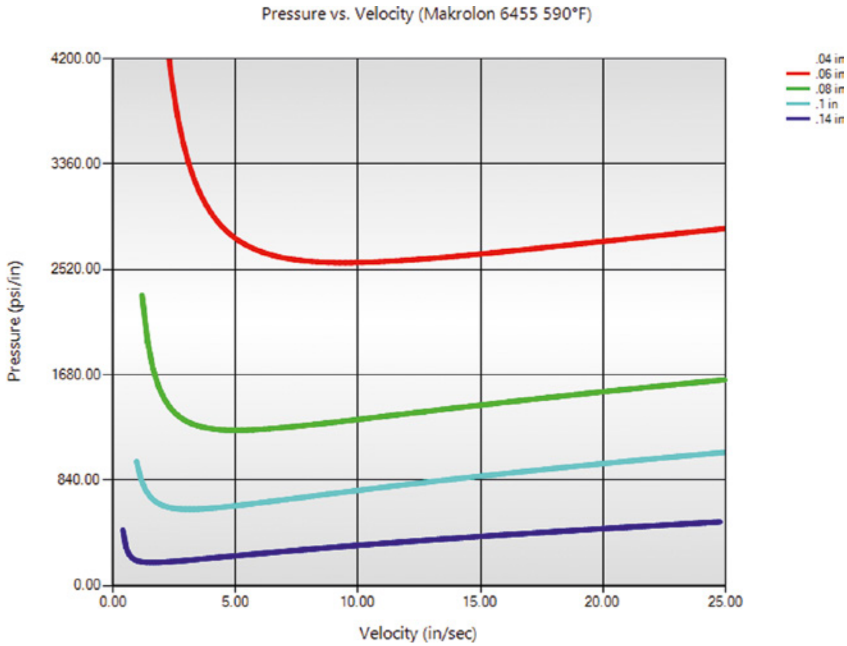


Figure 3.6 Mold filling pressure vs. flow front velocity at four different wall thicknesses (0.06", 0.080", 0.10", and 0.120")

Figure 3.7 contrasts the same PC as above to a PC/ABS at a wall thickness of 0.100" using the Moldometer. Note that increased shear thinning attributes of the ABS in the PC/ABS alloy decreases rate of the pressure rise at the faster fill velocities.

Figure 3.8 contrasts viscosity vs. shear rate data developed from a traditional capillary rheometer vs. the moldometer. Unlike a traditional capillary rheometer, the boundary of the moldometer is cooled to the same mold temperatures used during conventional injection molding. Therefore, the moldometer data includes the effect of the melts thermal exchange with the mold, including the development of a frozen layer. At the high shear rates, frictional heating is dominate with all wall thicknesses resulting in the viscosity data for all wall thicknesses beginning to converge. At these higher shear rates the frozen layer is minimized and therefore the data also begins to closely match the conditions measured in a traditional heated die capillary rheometer. However, at decreasing shear rates, the influence of the cold mold on melt temperature and a growing frozen layer can be seen. At these lower shear rates, a thin walled part is more heavily influenced by developing frozen layer than a thicker wall part. Also, at these lower shear rates we can see how differently a melt actually behaves in a mold vs. the conditions developed in a traditional capillary rheometer. Note that the viscosity data from the moldometer is not available at the lowest shear rates as the plastic material will freeze due to insufficient shear heating to offset heat lost to the cold mold.

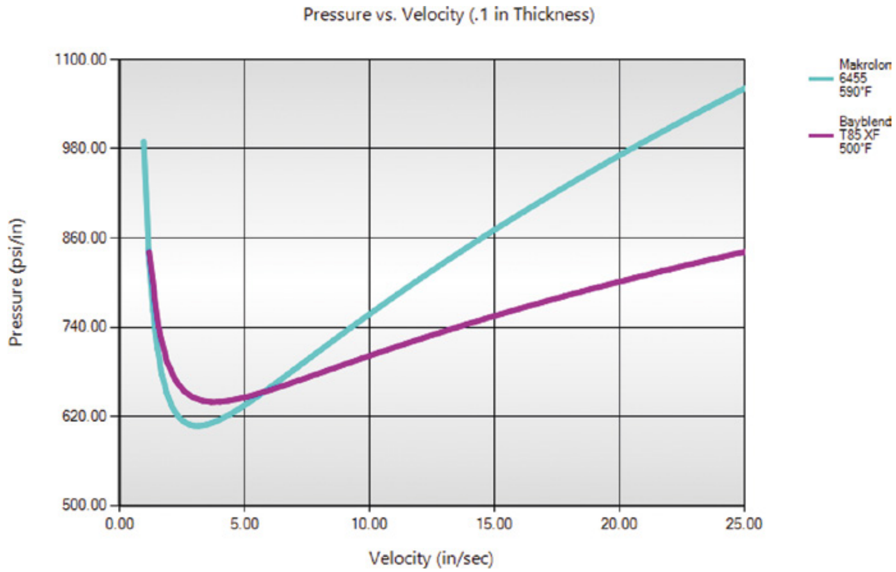


Figure 3.7 Contrasting the influence of injection rate on a PC versus a PC/ABS

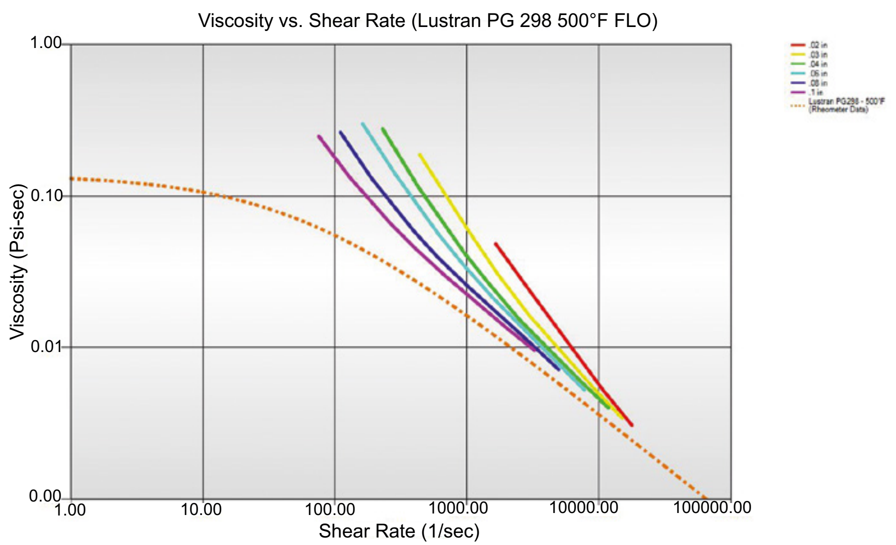


Figure 3.8 Viscosity vs. shear rate characteristics of a polymer when characterized in a conventional rheometer vs. how a polymer behaves when flowing through cooled channels .02 in (red), .03 in (yellow), .04 in (green), .06 in (blue), .08 in (violet), .1 in (purple), Lustran PG298-500°F (Rheometer Data) (dotted)

the runner and machine nozzle pressure was 10,800 psi. As the machine was capable of 20,000 psi, the high pressure loss in the runner did not create a problem.

It should be obvious now that performing a mold filling analysis without considering the nozzle and runner system could result in significant misjudgments about the ability to fill a part.

5.4 Use of Mold Filling Analysis

Injection mold filling analysis programs by companies like Autodesk Moldflow Inc. and Core-Tech Systems Co. provide an excellent tool for sizing runner systems. These programs provide information on pressure, melt temperature, and shear rate at various fill rates. Though shear rate can be determined using simple hand calculations, fill pressure and melt temperature at various fill rates require much more sophisticated solution methods and detailed characterization of the polymer. Of particular interest to most molders is determining if their mold will fill with a given runner and gate design and a given gating location on their part. To determine this, the melt delivery system and the part forming cavity must be modeled. To size runners, a skilled analyst does not require a detailed model of the cavity. Often they can use simplified geometries that represent the volume of the cavity and a flow length and thickness representative of the most difficult flow path through the cavity [4]. Early 2-D injection molding simulation programs used this method successfully for years. The advantage of this older 2-D method is that the modeling and analysis can take as little as a half an hour for a skilled analyst. These programs used a simple 1-D beam for runners, and although they did not provide any graphical feedback, they did provide good information on pressure, temperature, shear rate, and shear stress on the melt during mold filling. The risk of this technology originated mostly in poor application by the user. The modeling of the part required good interpretive skills and good ability to realize what the program could and could not provide.

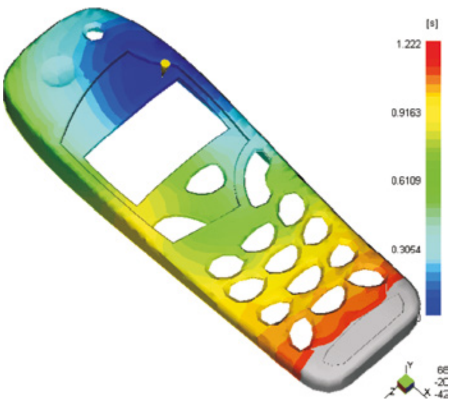


Figure 5.8 2½-D mold filling analysis output of fill pattern

Most of these early programs have been replaced by much more sophisticated 2½-D and 3-D programs that can provide much more detailed information on flow through the cavities (see

Figure 5.8). Detailed information on cavity conditions can be provided in easy to interpret colorized contour plots. Though these new programs present the impression that they are easier to use, they are significantly more complicated and compute-intense. They still require a skilled analyst to assure that the geometry and mesh is representing the critical regions to be analyzed. If sizing a runner and evaluating a gate design are the issue, these programs can be an over-kill and a waste of engineering time. This is particularly the case when many analysts still use the same 1-D beams to represent their runners as the older 2-D programs. The primary advantages of the newer programs are studying the filling patterns and melt conditions throughout a cavity and for the further analysis of mold cooling, part shrinkage, warpage, and structural performance.

Some cautionary remarks regarding the use of any of the standard 1-D, 2-D, 2½-D, and 3-D injection molding programs:

1. Mold filling analysis can provide good information on how small a runner can be while still allowing the mold to fill. With a cold runner, be careful that the size provided from a mold filling analysis is not too small to allow for the cavity to be properly packed out during compensation/packing phase. It is generally expected that the cold runner diameters should be no less than 1.5 times larger than the thickness of the part. Smaller diameters are possible but are more prone to packing issues. (Part requirements and design must be considered.)
2. One should be careful when trying to analyze an insulated or internally heated hot runner system. Most programs do not calculate the development of a frozen layer in these applications. Check with the software provider on how these conditions are handled.
3. The 1-D beams used in the 2-D and 2½-D filling analysis programs cannot pick up the shear-induced filling and melt imbalance in multi-cavity molds. Therefore, they also will not be able to pick up their influence on the part's shrinkage, warpage, and residual stresses.
4. At this time, all of the newer 3-D filling analysis programs struggle to predict the magnitude of the shear-induced filling and melt imbalances in multi-cavity molds (see Chapters 6 and 7 for details on shear induced melt variations developed in runners). Without careful meshing, these programs may only predict a small fraction of the melt variation and the influence it has on the part. Filling imbalances of less than 5% are often being predicted where the actual imbalance may be over 30%. Intra-cavity influences on filling patterns, shrinkage, residual stresses, and warpage are also commonly under-predicted.
5. Mold filling analysis is commonly used to artificially balance the filling of a fishbone type runner layout. These programs can significantly reduce the effort required to manually balance these molds. However, a molder should realize that an artificial filling balance will not balance melt condition, shrinkage, warpage, or packing.

Method 2: Method 2 solves the pressure through the annular gap without having to derive an equivalent rectangular shaped flow path.

$$Given: \eta = m\dot{\gamma}^{n-1};\ m = 0.179\,psi \cdot \sec;\ n = 0.681$$

$$\dot{\gamma}_{AnnularFlow} = \frac{6\,Q}{\pi(R_{Bore} + R_{Heater}) * (R_{Bore} + R_{Heater})^2}$$

$$= \frac{6 \cdot 2}{\pi(0.4 + .3125) * (0.4 - 0.3125)^2} = 697\sec^{-1}$$

$$\eta = m\dot{\gamma}^{n-1} = 0.179 \cdot 697^{0.681-1} = 0.0222$$

$$\Delta P_{AnnularFlow} = \frac{12Q\,\eta\,l}{\pi(R_{Bore} + R_{Heater}) \cdot (R_{Bore} - R_{Heater})^3}$$

$$= \frac{12 \cdot 2 \cdot 0.0222 \cdot 10}{\pi(0.8 + 0.625) \cdot (0.8 - 0.625)^3} = 3{,}526 psi$$

Note that in the above examples, pressure drop as determined by both methods are essentially the same. Also note that the pressure drop through the annular flow channel is nearly 8 times that found in an equivalent full-round flow channel. In actual applications, this will vary as the frozen layer development along the outside diameter of the internally heated annular channel is not considered.

5.5.2.1 Flow through a Hot Runner vs. a Cold Runner

For the most part, the pressure development in the runner system is the same for hot and cold runners. Both types of systems experience laminar flow and fountain flow, which means there is no flow at the mold wall. In other words, there is no slip of the melt at the wall of the mold as the plastic is being injected.

Hot runner molds typically have slightly larger diameter runners because there is no concern with runner regrind or concern with its cooling time. These larger diameters allow for reduced pressure drops through the runner. Despite the surrounding cold mold in a cold runner, the bulk temperature of the melt is very similar in both hot and cold runner systems due to the significant shear heating developed in a runner. This shear heating also minimizes the development of a frozen layer during mold filling in a cold runner.

5.5.3 Runner Effect on Cycle Time

5.5.3.1 Cold Runner and Sprue Cooling Time

The cooling time of the sprue and runner has the ability to affect the overall cycle time. Although the sprue and runner do not have to be frozen completely, they must cool long enough that they may be easily ejected. This rarely becomes an issue unless when molding thin walled parts. If the sprue puller region, which is normally the thickest area in the melt delivery system, is forcing the cycle time to be extended, a hot sprue may be a good replacement.

5.5.3.2 Hot Runner

Hot runners have a clear advantage over cold runners in most high speed thin walled molding applications. Time is saved as less material must be plasticated and injected to fill the runner, clamp stroke is reduced, runner ejection time and handling are eliminated, as well as eliminating additional cooling time that might be required for the cold runner. However, the hot runner can potentially extend cycle time in some cases, as it not only adds heat to the mold but restricts the location of cooling channels. This is particularly true in the gate region. Here the hot drop reaches directly to the part. The addition of cooling to this area is physically obstructed by the hot drop itself. Though cooling can be designed and machined in special channels around the drop tip, this is commonly left out by the designer due to cost and complexity. In addition, direct gate cooling can potentially cause premature gate freeze.

5.5.4 Constant Diameter vs. Graduated Diameter Runners

It is common practice, with geometrically balanced runners, to decrease the runner diameter at each branch as it progresses from the sprue (see Figure 5.12). This is a practice that is often blindly performed without understanding its purpose, or the potential negative effects.

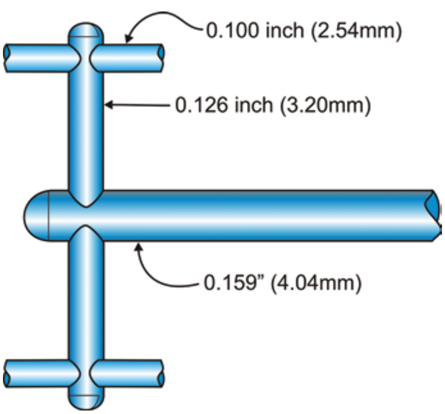


Figure 5.12 A graduated runner showing progressively increasing diameters from the tertiary to secondary to primary runner sections

When sizing a cold runner, its minimum diameter must allow for proper packing of the part. Therefore, if a runner is to have progressive runner branches with varying diameters, it must be designed from the gate back to the sprue. The smallest diameter runner would be attached to the gate and each successive branch back-toward the sprue would be increased.

6.3.4.2 Core Deflection

Core deflection is caused by unbalanced pressures developed from the melt on a core. The location of the gate has a significant impact on core deflection. Figure 6.28 shows two cores with three different gating locations. Gate locations 1 and 2 will both result in high pressure developing on the side of the core near the gate. This will cause the core to bend away from the gate. Gate location 3 is preferred when gating concentric parts. Not only will gate location 3 reduce the potential for core bending, it should also help prevent air traps, weld lines, and non-concentricity. However, despite this apparently ideal center gating location, filling patterns in center-gated parts in multi-cavity molds are almost always unbalanced. Shear-induced melt variations again will create side-to-side filling and packing variations, which can deflect the mold core forming the part.

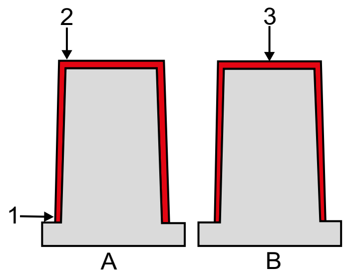


Figure 6.28 Gating locations 1 and 2 will cause core deflection. Gating location 3 should not contribute to core deflection as long as the melt entering the cavity has symmetrical temperature and shear conditions. However, if fed by a traditional 2nd generation branching runner (2 or more cavities), cavity filling will be unbalanced

Figure 6.29 illustrates the development of a side-to-side filling variation that can develop in a simple four-cavity, three-plate cold runner or hot runner mold. The highly sheared laminates, developed from the machine's nozzle and sprue, are split at the primary runner. This creates a bottom to top (sprue side to core side) melt variation in the primary runner, which continues into the part forming cavity. This can potentially deflect the core during both the filling and the packing stages and result in variations in wall thickness within the part. This wall thickness variation can then cause the part to warp. The resulting wall thickness variations and warpage can often be traced to be directly related to the expected position of the high and low sheared materials. Interestingly, it is often found that the actual core deflection is away from the low sheared material side of the core. This is analogous to the condition where the last filling cavities in an unbalanced mold can sometimes end up producing the largest and heaviest parts.

Even if the core does not deflect, significant problems can develop from the melt variations entering a cavity. Figure 6.30 shows a small, center-gated canister molded in a 16-cavity hot runner mold. Despite the ideal center gating location, a significant filling imbalance can be seen. The lead flow on the side of the part is fed from the high sheared regions of the runner. The flow in this case actually races down one side, around the flange at the open end, and creates a gas trap along the side of the part.

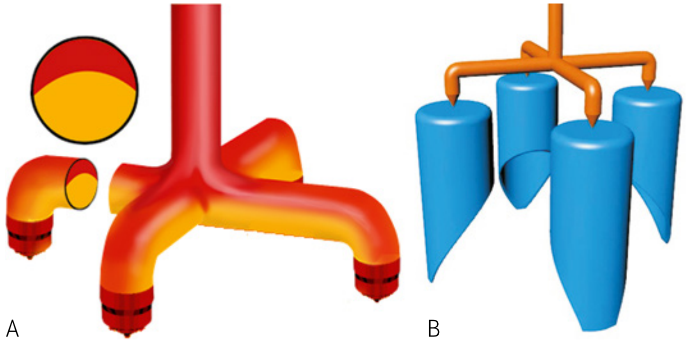


Figure 6.29 Top figure (A) illustrates the development of asymmetric melt conditions in a simple 4-cavity mold. The bottom figure (B) illustrates the potential intra-cavity filling imbalance that can be expected to result in core deflection and side to side shrinkage variations causing warpage

Figure 6.30 Center-gated canister molded in a 16-cavity hot runner. Asymmetric melt conditions resulted in the intra-cavity filling imbalance shown

6.3.4.3 Effect on Concentric Parts (Gears, Fans, and Others)

The continuation of unmanaged shear-induced melt variations into any centrally-gated part can create significant challenges that are commonly misunderstood. This is particularly the case with high precision parts such as gears and fans. Both of these require excellent concentricity. Figure 6.31 is an illustration based on an actual industrial automotive case of a large fan produced in a two-cavity, hot-to-cold runner system. Each drop of the two-drop hot runner is feeding a wagon wheel cold runner with 10 spokes, each directly feeding the fan. Despite the perfectly geometrically balanced runner system, each cavity was filling eccentrically. The half of each cavity toward the edges of the mold was filling before the half in the center of the mold. The resulting eccentric filling and packing caused a disabling imbalance in the finished molded part. The part weight imbalance was severe enough that the part had to be hand balanced using weights following molding. Initially, it was thought that the mold's cores were deflecting outward from the mold, thus opening the flow channel and reducing the pressure drop in those areas. However, it was found that when one cavity was shut off, the parts filled

8.5 Effects of Gate Diameter in Multi-Cavity Molds

Mold filling imbalances in multi-cavity molds are particularly sensitive to variations in gate sizes. Even gates designed within common machining tolerances can result in unexpected and undesirable filling imbalances. As the cross sectional size of the gate decreases, the process becomes more sensitive to slight variations in the gate diameter size.

Through use of the flow grouping and mold balance diagnostics method presented in Section 15.1, the impact of mold filling imbalances, as effected by dimensional variations in mold steel, can be isolated and quantified. Using this method in numerous commercial applications, it was found that significant mold filling imbalances could be attributed to very small variations in a gate diameter. Often, these imbalances were occurring despite the fact that the gates were sized within the designer's tolerances. Gates on the high versus low end of the tolerance were a common source of the problem. The problem has been observed in both hot and cold runner molds. This led to a couple of studies using mold filling simulation to help isolate and quantify the effect of gate diameter variations as compared to the resulting filling imbalances. As the purpose of the study was to evaluate gate size influences, simple 1D beam runners and gates were used in order to eliminate any influence of shear induced melt variations.

8.5.1 Study 1

The first study was to evaluate the effect of small changes in gate diameter on pressure. The changes were based on tolerances that might be considered very tight to fairly loose. These tolerances are ± 0.005 mm (0.0002 in.), ± 0.0127 mm (0.0005 in.), ± 0.0254 mm (0.001 in.), and ± 0.05 mm (0.002 in.). The high and low limits of these tolerances were applied to a 0.762 mm (0.030 in.) inch long gates with diameters of 0.51 mm (0.020 in.), 1.02 mm (0.040 in.), 1.52 mm (0.060 in.), 2.03 mm (0.080 in.), and 2.54 mm (0.100 in.). The results are summarized in Table 8.1.

As seen in Table 8.1, the smaller the gate, the more significant the impact of variations in gate diameters. With a 2.54 mm (0.100 in.) diameter gate, a variation of ± 0.0254 mm (0.001 in.) has an 8% effect on pressure, whereas the same tolerance on a 0.51 mm (0.020 in.) diameter gate will have a 49% effect on pressure. These small diameter gates are commonly used in high tolerance parts, including those used for manufacturing electrical connectors.

Table 8.1 Pressure Variation Based on Gate Diameter Tolerance (Dimensions in mm)

	Tolerance			
	± 0.00508	± 0.0127	± 0.0254	± 0.0508
Gate Dia	% P Var	% P Var	% P Var	% P Var
0.508	8%	22%	49%	123%
1.106	4%	11%	22%	49%
1.524	3%	7%	14%	31%
2.032	2%	5%	11%	22%
2.54	2%	4%	8%	17%

8.5.2 Study 2

The second study looks at the effect on mold filling imbalance in an eight-cavity geometrically balanced cold runner mold. Again mold filling simulation was used. The part was a simple flat plaque having a volume of 2.419 cm^3 (0.1476 $in.^3$). The runner had a standard round channel with a 3.175 mm (0.125 in.) diameter. The parts were gated using a pinpoint gate with a 0.762 mm (0.030 in.) length. Three gate diameters were used, 0.762 mm (0.030 in.), 1.02 mm (0.040 in.), and 1.27 mm (0.050 in.). The gate diameters of the four inside and four outside cavities were varied to the upper and lower limits of a specified tolerance and the results were analyzed. The tolerance values that were used are; ± 0.0127 mm (0.0005 in.), ± 0.0254 mm (0.001 in.), ± 0.05 mm (0.002 in.), and ± 0.102 mm (0.004 in.).

Filling analyses were performed, using Moldflow's MPI 6.0 software, running DuPont's Zytel nylon and GE's Cycolac ABS, with a set injection time of 1 second for all simulations. The gate diameters for flow group #2 (outside cavities) were set to the upper limit of the tolerance, where the gate diameters for flow group #1 (inside cavities) were set to the lower tolerance limit. Comparisons between the flow rates through the gates of the different flow groups were made, and a percent difference was calculated to find the percent flow imbalance. The flow rates directly correspond with the fill time of the cavities.

As with Study 1, it was found that the smaller the gate, the greater the impact of varying gate diameter. For instance, a 0.762 mm (0.030 in.) gate will see more of a percent imbalance over its tolerance range with varying gate diameter than a 1.02 mm (0.040 in.) diameter gate. As well as the larger the tolerance the greater the imbalanced experienced. The graph in Figure 8.43 shows several important factors of gate size and variation in the nominal gate size. The y-axis represents the percent imbalance, where the x-axis represents the tolerance limit set for the specific gate. For example, varying the 0.762 mm (0.030 in.) gate to the upper and lower ends of the ± 0.0254 mm (0.001 in.) tolerance limits (gate diameters of 0.7874 mm (0.031 in.) and 0.737 mm (0.029 in.)) resulted in an imbalance of nearly 17% between the two flow groups. This reinforces the idea that small deviations from a nominal gate size can and will have significant effects on the flow imbalance and overall process window. As the deviation from the nominal gate diameter increases, the percent imbalance will grow.

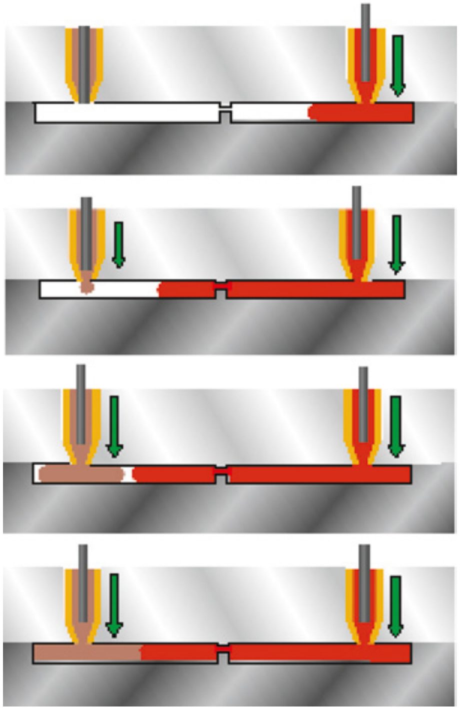


Figure 11.24 Use of sequential valve gating to assure good strength across the hinge and good packing on either side of the hinge

11.3.5.3 Valve Pin Movement Control for Sequential Gating

Until recent years, most all valve pin movements have been limited to providing a single fast speed open, to a fully open position, and a single fast speed close to a full close position. This type of control can work well in many applications. However, when using cascading sequential valve gating, when progressive gates are opened during mold filling, a sudden change in flow front velocity results that can often cause flow mark on the surface of the part. This is particularly acute with glossy part surfaces.

In response to this issue, hot runner manufactures have been developing systems that provide a higher level of control of the opening stroke, and some also proving a similar high level of control to the closing stroke. Essentially these systems can profile the opening and closing stoke of the pin rather than the more conventional one fast speed to one position open and one fast speed to one position closed, with all pins set to the same high speed. The ability of the molder to profile the opening stroke allows them to discretely control the introduction of melt from each progressive gate in order to eliminate surges in flow front velocity, and thereby address the resultant cosmetic issues.

Figure 11.25 is a plot of an automotive part being fed by five gates using cascading sequential valve gates. The left side of the top figure, and the corresponding close-up bottom left, show the flow lines resulting from traditional quick full open valve gates. This is contrasted to the right side of the top figure, and corresponding close-up on the bottom right where the valve opening has been profiled using a servo driven valve gate to eliminate the flow lines.

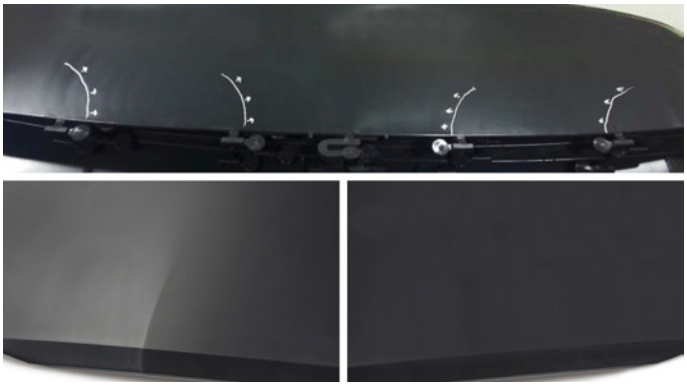


Figure 11.25 Automotive part where the left side was molded with a traditional fast-opening valve gate versus the right side that was molded using a profiled servo driven valve gate (Courtesy: HRSflow)

The newer systems with the more controlled opening and closing motions are still mostly based on hydraulic drives. However there are also electronic servo driven and pneumatic systems. With hydraulic and pneumatic systems, speed can be controlled by manually adjusting flow control valves. The valves can be located downstream of the pin's hydraulics (return to tanks side of the circuit) in order to maintain positive control of the pin movement. This type of system may only provide a single slowed opening stroke of the valve pin to a single fully open position, while some may have further profiling capabilities. In application, an operator may progressively throttle down the speed of an opening gate while observing a pressure versus time curve and comparing this to the visual inspection of the part. Once the cosmetic issue has been eliminated at the second gate, the operator would then repeat the process at each of the progressive gates until their objective has been met. Figure 11.26 is an illustration of the valve gate opening and a corresponding valve pin position (y-axis) vs. time (x-axis) achieved with a controlled slow speed hydraulic system during the opening stroke. The controlled time to open is established as described above.

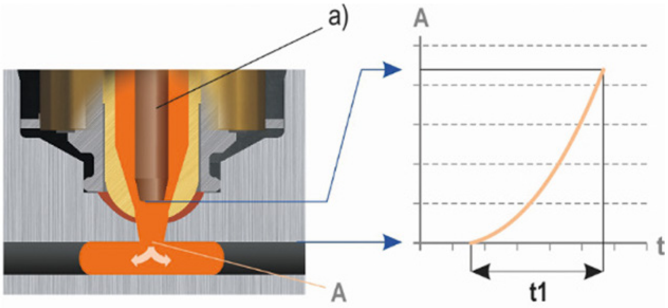


Figure 11.26 Controlled valve pin (a) opening with corresponding response curve of position (y-axis) vs. time (x-axis) (Courtesy: INCOE)

Figure 15.22 Record grooves or orange peel

Possible causes	Possible remedies
Mold build up or deposits	Check for residue or deposits on the mold/cavity surface. If there are mold deposits, see "Mold buildup."
Mold surface finish	Check surface of cavity for proper polish or finish and whether it is clean. Repair and clean.
Slow filling	Increase injection rate, this decreases resin viscosity and allows more pressure to be transferred to the cavity. If 1st to 2nd stage switchover is < 0.1 s, ensure velocity is not pressure-limited.
Low cavity pressure	Increase 2nd stage pressure. Increase 2nd stage time, and if possible remove the same amount of time from the cooling or mold closed timer to keep cycle time constant.
Mold temperature	Increase mold temperature. Decrease mold temperature.
Melt temperature	Check melt temperature, adjust to within the manufacture's guidelines if temperature is outside limits. Try higher end and lower end of resin supplier's guidelines.
Uneven filling of a single cavity	Balance flow path with flow leaders if possible. Increase injection rate.
Unbalanced filling in multi-cavity molds	Adjust runner size to balance filling. Do not adjust gate size to balance filling; this will provide various gate seal times and vary part dimensions, weight, etc.

Pinking of the Part

Relatively rarely it happens that parts will turn pink while in storage. The cause is usually carbon monoxide gas reacting with components of the plastic.

Possible causes	Possible remedies
Carbon monoxide	Minor amounts for carbon monoxide are known to discolor certain resins. Remove parts to open area and see if discoloration disappears. Exposing the part(s) to sunlight can accelerate the disappearance of the discoloration. If discoloration reverses, remove all gas fueled lifts etc. from storage area. Improve storage area ventilation. Go to battery operated fork lifts.

Pitting

Possible causes	Possible remedies
Trapped gases dieseling	See "Burns." If it is due to dieseling, do not run the mold, further damage will result.
Corrosion or chemical attack by the resin or additive on the steel	Check for resin compatibility with the steel of the mold. If acid gases are possible, a more chemically resistant surface may be required. A different steel or coating of the existing surface should be specified.
Abrasive wear, erosion	Highly filled resins can pit and erode a mold's surface finish. Change gate location, coat cavity with a wear resistant finish. Rebuild tool with appropriate hardened steel.

Poor Color Mixing

See also "Color Mixing"

Race-Tracking, Framing, or Non-Uniform Flow Front

The flow front should be a continual half-circle fill from the gate.

Possible causes	Possible remedies
Non-uniform wall thickness	Thicker sections of part fill preferentially due to lower melt pressures required to fill. Plastic flow will accelerate in thicker sections and hesitate filling a thin section. This may allow the plastic to "race-track" around the perimeter or section of a part and trap air or volatiles. Try faster injection rates but it is unlikely this will solve the problem as you are fighting a law of physics. Round the edge or taper the junction between the nominal wall change. The correct fix is to redesign with a uniform nominal wall.
Gate location	Gate into the thick area and provide flow leaders to the thin areas to provide uniform filling.
Hot surface or section in the mold	Allow the mold to sit idle until mold is at uniform temperature. Make and save first shot for 99% full. If flow path is different than in later shots, it is a tool-steel temperature and cooling issue. Check mold for hot spots. Get uniform cooling.

Record Grooves, Ripples, Wave Marks

These are concentric grooves or lines usually at the leading edge of flow. The flow front is hesitating, building up pressure then moving a short distance and hesitating again. This is almost always related to lack of adequate pressure at the flow front or slowing of injection velocity.

Possible causes	Possible remedies
Pressure limited 1st stage or lack of velocity control	Double check that the pressure during 1st stage is 200–400 hydraulic psi lower than the set first stage limit. Make sure there is enough pressure differential (delta P) between the highest pressure during 1st stage and the set pressure limit for 1st stage. First stage pressure limit should be higher than the pressure used during 1st stage.
Incorrect position transfer	Take 2nd stage pressure to 300 psi plastic pressure or if the machine does not allow this, take 2nd stage time to zero. The part should be 95–99% full. If this is a thin-walled part, the part should be full with only slight underpack near the gate. Adjust position transfer to provide appropriate fill volume.
Melt temperature too low	Check melt temperature via the hot probe technique or appropriate IR sensor. Make sure it is within the resin supplier's recommended range.
Poor 1st to 2nd stage switch-over response	Note response of hydraulic pressure at switch-over. It should rise to the transfer point, then drop rapid to the set 2nd stage pressure. If hydraulic pressure drops much below set 2nd stage pressure, the flow front may be hesitating and building a high viscosity. Repair machine.
Low pack rate or volume	Increase pack rate or volume of oil available for 2nd stage.
Low mold temperature	Increase mold temperature 20–30 °F. Decrease cycle time. This will raise steel temperature in the mold.

Screw Recovery, Slow Recovery, Screw Slips or Does Not Feed

The metering section of the screw pumps plastic forward, which pushes the screw back.

Possible causes	Possible remedies
Feed throat temperature	Run throat temperature at 110 to 140 °F for most resins. For high-end engineering resins you may want to go higher. Do not run feed throat at 60–80 °F. Feed throat should be PID temperature-controlled.
Feed problems	Check size of granules and flow through hopper and feed throat. Ensure that material gravity-feeds correctly when resin is being loaded into the hopper. Vacuum loading may interrupt normal gravity-feeding, especially with single shot loaders. If coloring at the press, check recovery without colorant. Certain color concentrate carriers can increase recovery times, too much wax or oil.
Heavily carbonized or blocked flights	Standard general purpose screws are notorious for dead spots behind flights. These can have large carbon or other deposits that block plastic flow. Check screw for clean polished flights.
Worn screw and/or barrel	Worn screws and barrels will provide better mixing but slow recovery rates as plastic back flushes over flights. Flights should be sharp, screw root should be highly polished, no nicks or scratches.
Moisture	Check moisture content of plastic, check feed throat for cracks leaking water.
Granule size	Plastic granules should be uniform in size and shape. A wide range in granule size, fines and small granules along with large chunks of regrind will cause feeding problems. This includes large and small pellets in virgin.
High back pressure	Target 1,000 to 1,500 psi melt back pressure. Try lower back pressure.

Possible causes	Possible remedies
High RPM	Try lower screw rotate speeds; better melt uniformity and mixing are obtained with slow screw speeds. Use all but ~ 2 seconds of the cooling time for plasticating. Do not lengthen the cycle.
Incorrect barrel temperature settings	Start by setting front and center zones to the center of the resin suppliers recommended range. Set rear zone at the minimum of the range. Back pressure set at 1,000 melt psi. Average recovery time for 10 cycles. Repeat with rear zone 10 °F higher until you have reached the rear zone setting at the maximum recommended by the resin supplier. Pick the temperature that gives you the minimum recovery time.
Poor screw design	See “Color mixing” and “Screw design”

Screw Slip

See also “Screw Recovery”

Screw Design

Possible causes	Possible remedies
Standard general purpose screw	These are known to produce unmelt due to solids bed break up. They should be replaced with melt-uniformity screws: Minimum L/D is 20/1. This is an industry problem; most (99%) of general purpose screws do not provide uniformly melted polymer coming out of the nozzle.
Screw and barrel metals	Recommend: bimetallic or hardened barrels and soft screws like stainless steel. Chemically resistant screw material is especially critical for clear resins. Screw should be polished with sharp edges on flights with the back of the flights rounded with a large radius to prevent dead spots and carbon buildup. Screw root and flight channels should be highly polished with no nicks or scratches. A modified barrier should lead from the transition zone to the metering zone.
Barrier flights	Generally not recommended, unless short and at the end of the transition zone or beginning of the metering section. Often cause severe degradation and overheating.
Vented barrels	Vented barrels, though uncommon, do have their purpose. They provide excellent melt uniformity and process resins that are not subject to hydrolysis more uniformly. Unfortunately, their design is often poor. The two-stage screw must be designed with a continuous flight through the decompression section. The first stage should be cut such that it cannot overpump the second stage. Vented barrels require near zero back pressure to prevent vent flooding. This presents purging and residence time problems. See Figure 15.10, Figure 15.17, and Figure 15.23.

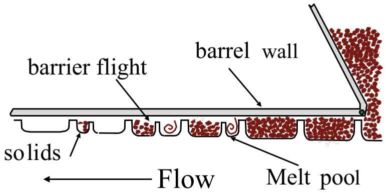


Figure 15.23 Barrier screw

Shorts or Short Shots or Non-Fill

Part is short or some section of the part, such as a rib is not completely filled out.

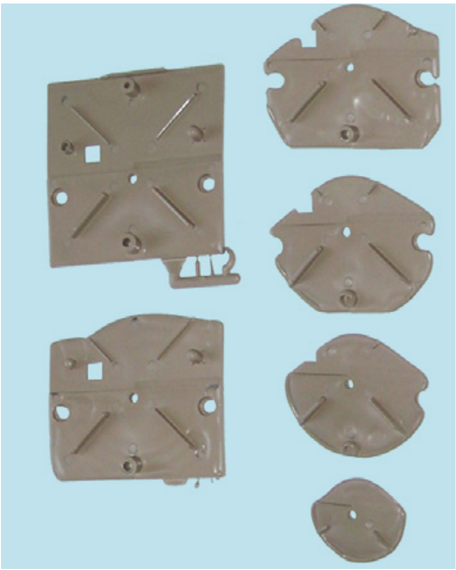


Figure 15.24 Short shot series filling a part; the part is center-gated

Possible causes	Possible remedies
Consistent short shots	
Incorrect shot size	Take 2nd stage pressure to 300 psi plastic pressure or lower ; if the machine does not allow this, take 2nd stage time to zero. The part should be 95–99% full. If this is a thin walled part, then the part should be full with only slight under-pack near the gate. Adjust position transfer to provide appropriate shot size or 1st stage volume. Do not use 1st stage pressure-limit to adjust the amount of plastic that enters the cavity on 1st stage.
Pressure-limited 1st stage or lack of velocity control	Double check that the pressure during fill or 1st stage is 200–400 hydraulic psi lower than the set first stage limit. Make sure this is enough pressure differential, delta P, across the flow control valve.
Injection rate	Increase injection rate to decrease viscosity.
Non-return valve or barrel worn or broken	Check non-return valve and barrel, are they in specification? If the non-return valve is OK, double check barrel for wear and ovality. Repair or replace as needed. Note: non-return valve should not have mating angle between seat and sliding ring. This should be a stepped angle for positive shut off.
Large pressure drop	Perform a short shot analysis for pressure loss. Note pressure at transfer for shots making: 1) 99% full part, 2) Sprue, runner and gate, 3) sprue and runner only, 4) purge full shot through the nozzle into the air. This is best done using one velocity during first stage. Use intensification ratio to calculate pressure drop for a) nozzle, acceptable range 200–4,000 psi b) sprue and runner, acceptable range 200–5,000 psi c) gate, acceptable range 200–5,000 psi d) part, acceptable range 200–40,000 psi. Evaluate where largest pressure drop is and remedy. A restriction of flow will be discovered with this method.

Possible causes	Possible remedies
Trapped gas or air	Progressively short shoot the mold using shot size or position transfer to make 10, 20, 40, 60, 70, 80, 95% full parts. Note if there is any sign of the plastic flow front coming around on itself. Is there a racetrack effect? This includes jetting. Note if ribs are covered before completely filling. This test cannot be done correctly if you reduce 1st stage pressure or velocity. You must be velocity-controlled for 1st stage. The concept is to find where the gas is coming from and eliminate its source. Vent the tool properly or use porous steel to eliminate gas traps. Jetting can cause gas and air to be trapped. Check for moisture, steam is a gas. Change gate location. Pull a vacuum on the mold cavity during fill.
Insufficient 2nd stage pressure	Make sure 1st stage is at the right shot volume; part should be 95–99.9% full at the end of 1st stage. If OK, raise 2nd stage pressure.
No cushion	Ensure adequate cushion to allow for transfer of packing or 2nd stage pressure.
Melt temperature	Verify that melt is within the resin supplier's recommended range.
Mold temperature	Try higher mold temperatures and or faster cycle times.
Resin viscosity	Change resin to a higher melt flow rate. Be careful as properties may decrease due to lower molecular weight. Parts must be fully tested in the application for correct performance.
Long flow length	For every mm of flow there is a pressure drop. Long flow lengths have large pressure drops. Add gates or flow leaders. Last resort increase nominal wall.
Check balance if a multi-cavity tool	See text for the Flow Grouping mold diagnostics process for determining mold balance.
Thin nominal wall	Add flow leaders if possible. Increase nominal wall if all other avenues fail. Thicker nominal wall will reduce pressure loss.
Intermittent short shots	
Non-return valve or barrel worn or broken	Check non-return valve and barrel, are they in specification? If the non-return valve is OK, double check barrel for wear and ovality. Repair or replace as needed. Note: non-return valve should not have mating angle between seat and sliding ring. This should be a stepped angle for positive shut-off.
Cushion not holding	Note cushion repeatability, if varying by more than 0.200 in. or 5 mm, check non-return valve as above. Try a larger decompression stroke to help "set" the check valve. Be careful not to suck air into the nozzle and cause splay.
Contaminated material	Check gates and parts for foreign material. Check quality of regrind. See introduction to "Color mixing."
Check balance if a multi-cavity tool	See text for Flow Grouping method for determining mold balance.
Melt temperature	Verify that melt is within the resin suppliers recommended range.
Mold temperature	Try higher mold temperatures and or faster cycle times.
Unmelt	Look for unmelted granules in the part, color streaks, see "Screw design" and introduction to "Color mixing". Provide uniform melt to the gate.
Cold slug	Occasionally it is possible for a cold slug from the nozzle to go beyond the sucker pin and plug a gate. Check nozzle for cold slug formation. See "Nozzle drool."
Insufficient 2nd stage pressure	Raise 2nd stage pressure.
Trapped gas	See above, see also "Bubbles."

Index

N

O

P

T

U

V

W

X

Z