Moldflow Design Guide

A Resource for Plastics Engineers

First Edition

Edited by Jay Shoemaker
Moldflow Corporation
Framingham, Massachusetts, U.S.A

Moldflow Corporation
492 Old Connecticut Path, Suite 401
Framingham, MA 01701 USA
Tel: +1 508 358 5848
Fax: +1 508 358 5868
www.moldflow.com

Moldflow Design Guide, First Edition

Foreword

The drive toward fast, cost-effective, and reliable plastics manufacturing has been Moldflow's sole guiding goal since the company was founded over 25 years ago.

This focused determination led us to introduce many new and exciting tools into the market, each contributing to achieving our goal in some way, whether by driving cost out of production with reduced material usage or shortened cycle times, reducing mold delivery time by minimizing re-work, or increasing the reliability of supply by enabling higher quality products to be manufactured with greater surety in scheduling.

The artificially balanced, multi-cavity and family molds that are now commonplace were made practical through the advent of our early simulation and runner balancing capabilities, which were introduced in the late 1970s and early 1980s. As these tools evolved, we were able to visualize, and therefore control, flow patterns and weld lines. This evolution continued until we arrived in the 2000s with an array of sophisticated technology to control warpage, account for heat transfer, predict core shift, adapt to new molding processes, and much more. From traditional midplane technology to fully three-dimensional simulations, all our solutions are well integrated into a solid-modeling design environment.

As the technology has evolved, so has its usage. When Moldflow simulation technology was introduced, its primary purpose was to search for remedies to pre-existing molding problems. It soon became evident that the insight the software provided to solve molding problems would be better applied ahead of actual molding, during the design process. This methodology, which we call "problem avoidance," was the primary use for Moldflow technology for the first 20 years of its existence.

For Moldflow, this created a unique challenge: to open the world of manufacturing to the designers of parts and molds. What constitutes an ineffective design for molding may be apparent to a seasoned processing engineer looking retrospectively at a poorly performing tool, but how can design engineers use the CAE tools to visualize, diagnose and solve these same issues ahead of time—without 20 years of molding experience? How can manufacturers go further and use information that cannot be seen in the real molding process but is revealed via simulation?

The key that unlocked this puzzle began its life as the *Moldflow Design Philosophy.* This is widely viewed as the most important publication Moldflow has ever produced and has spawned follow-on works on related subjects. Rather than provide insight into the operation of the simulation tools, *Moldflow Design Philosophy* set forth simple principles that transcend any specific software application and, as a result, are as valid with today's advanced simulation products as they were over two decades ago.

In more recent years, another transition has occurred. The global imperative to drive down the cost of manufacturing has led to the use of molding simulation as a cost optimization tool rather than for problem avoidance. This change has increased the number of Moldflow users by an order of magnitude across a far broader cross-section of the plastics industry. Greater design-centricity leads to even more dependence on the plastics design principles, which can be used to drive optimization.

Despite a quarter of a century of technological advances, the golden years of CAE are ahead of us as our industry takes a broader and more integrated view of what it takes to manage a product's life cycle. Moldflow is proud of its contributions to date and will continue to focus on developing innovative technology coupled with practical design principles to deliver more profitable manufacturing.

Roland Thomas
President & CEO, Moldflow Corporation

Preface

About this Book

The origins of this book include not only *Moldflow Design Principles*, but also *Warpage Design Principles* published by Moldflow, and the *C-MOLD Design Guide*. Collectively, these documents are based on years of experience in the research, theory, and practice of injection molding. These documents are now combined into this book: the *Moldflow Design Guide*. The *Moldflow Design Guide* is intended to help practicing engineers solve problems they frequently encounter in the design of parts and molds, as well as during production. This book can also be used as a reference for training purposes at industrial and educational institutions.

How to Use this Book

This book has several chapters and appendices that deal with different stages of the design process and provides background on the injection-molding process and plastic materials.

- The first three chapters introduce injection molding how polymers flow inside injection molds and how molding conditions and injection pressure influence the process.
- Chapter 4 discusses Moldflow design principles and how they relate to making quality parts.
- Chapter 5 introduces the finite element mesh technology used by Moldflow and how these meshes influence the quality of the analysis.
- Chapters 6 to 9 introduce design concepts for the product, gates, runners, and cooling systems.
- Chapter 10 introduces concepts relating to shrinkage and warpage and how Moldflow is used to determine the amount of shrinkage and warpage a molded part will have and what causes the warpage.
- Chapter 11 discusses the design procedure for analyzing injection-molded parts.
- Chapter 12 discusses major part defects found on injection-molded parts.
- Finally the four appendices discuss basic injection-molding machine operation, process control, variants of the standard injection-molding process, and plastic materials.

Benefits of Using CAE

The injection-molding industry has recognized that computer-aided engineering (CAE) enhances an engineer's ability to handle all aspects of the polymer injection-molding process, benefiting productivity, product quality, timeliness, and cost. This is illustrated by a wealth of literature and the ever-growing number of CAE software users in the injection-molding industry.

CAE Predicts Process Behavior

Ideally, CAE analysis provides insight that is useful in designing parts, molds, and molding processes. Without it, we rely on previous experience, intuition, prototyping, or molding trials to obtain information such as polymer melt filling patterns, weld-line and air-trap locations, required injection pressure and clamp tonnage, fiber orientation, cycle time, final part shape and deformation, and mechanical properties of molded parts, just to name a few. Without CAE analysis, other equally important design data, such as spatial distributions of pressure, temperature, shear rate, shear stress, and velocity, are more difficult to obtain, even with a well-instrumented mold. The process behavior predicted by CAE can help novice engineers overcome the lack of previous experience and assist experienced engineers in pinpointing factors that may otherwise be overlooked. By using CAE analysis to iterate and evaluate alternative designs and competing materials, engineering know-how in the form of design guidelines can be established relatively faster and more cost-effectively.

User Proficiency Determines the Benefits of CAE

While CAE technology helps save time, money, and raw material, as well as cuts scrap, reduces the rejection rate, improves product quality, and gets new products to market faster, it is by no means a panacea for solving all molding problems. Rather, it should be recognized that CAE analysis is essentially a tool, designed to assist engineers instead of taking over their responsibilities or replacing them. Like many other tools, the usefulness of CAE technology depends on the proficiency of the user. The benefits mentioned above will not be realized unless the CAE tool is used properly. To be more specific, the accuracy of CAE analysis depends greatly on the input data provided by the user. In addition, the results generated by CAE analysis need to be correctly and intelligently interpreted by the user before sound judgments and rational decisions are made. Otherwise, users will simply be swamped by the vast amount of data without getting any useful information.

Acknowledgements

The *Moldflow Design Guide* would not have been accomplished were it not for the vision of Ken Welch. Ken and I have discussed the value of assembling the best of the *Moldflow Design Principles*, *Warpage Design Principles*, and the *C-MOLD Design Guide* into a single book for several years. With Ken's leadership, he gave the project to Steve Thompson's training group, of which I am a part. Steve helped me coordinate the resources necessary to get this project done. I could not have done this project without Steve's help and guidance.

A review of the content was part of the development of the *Moldflow Design Guide*. Moldflow developers including Peter Kennedy, Rong Zheng, Zhongshuang Yuan, and Xiaoshi Jin have reviewed sections of the book. Moldflow's application engineers and other technical staff with Moldflow have also reviewed sections. These reviewers include Chad Fuhrman, Matt Jaworski, Christine Roedlich, Eric Henry, Olivier Anninos, Paul Larter, and Ana Maria Marin. A special thanks goes to Mike Rogers, who reviewed the entire book for me and provided critical feedback on the content and organization of the book. I would also like to thank Kurt Hayden of Western Michigan University for reviewing the appendix on process control. His many years of experience of process setup and optimization was invaluable.

Finally, I would like to thank members of Moldflow's Technology Transfer group for supporting me in the writing of this book. These members include Marcia Swan, Caroline Dorin, Robert Ashley, Melissa Haslam, Darren Seamons, and Gregory Brown.

On a personal note, I would like to acknowledge and thank Paul Engelmann, Professor and Department Chair, Western Michigan University, Department of Industrial and Manufacturing Engineering, for being my friend and mentor during my career. With Paul, I have been able to teach and participate in research he has done on injection molding tooling and processing at Western Michigan University. I have found working with Paul has made me a better Moldflow user and engineer by providing another perspective on how Moldflow can be used to solve injection molding problems.

Jay Shoemaker, Editor

Contents

1 Polymer Flow Behavior in Injection Molds

- Phases of injection molding
- How do plastics flow?

1.1 Phases of Injection Molding

Any molder can prove that all the conditions and effects discussed in this chapter do indeed occur during the injection molding process. While this knowledge alone can somewhat improve quality, it is only with the use of Moldflow analysis during the initial design stage, with the mold designed for the optimum filling pattern, that these effects can be controlled and the full benefits obtained.

Flow technology is concerned with the behavior of plastics during the mold filling process. A plastic part's properties depend on how the part is molded. Two parts having identical dimensions and made from the same material but molded under different conditions will have different stress and shrinkage levels and will behave differently in the field, meaning that they are in practice two different parts.

The way the plastic flows into the mold is of paramount importance in determining the quality of the part. The process of filling the mold can be distinctly analyzed with the ability to predict pressure, temperature, and stress.

1.1.1 How Plastic Fills a Mold

This was investigated using a centrally gated mold shaped like a dinner plate with a thick rim around the outside as shown in Figure 1.1. It was found that the injection molding process, although complex, could be divided into three phases (we use the word *phase* to avoid confusion with injection stage, as used with programmed injection).

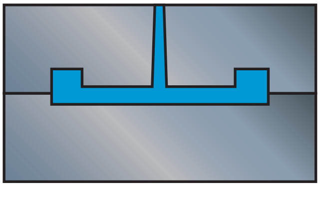


Figure 1.1 Cross-section of disk mold used to investigate flow

1.1.1.1 Filling Phase

As the ram moves forward, it first moves at a steady speed as the plastic flows into the cavity. This is the filling phase. This phase lasts until the mold is just filled. See Figure 1.2 and Figure 1.3.

1.1.1.2 Pressurization Phase

The pressurization phase begins when the ram moves forward after the filling phase to bring the mold up to pressure. When the mold is filled, the ram will slow down, but it still moves quite some distance because plastics are very compressible materials. At injection molding pressure, an extra 15% volume of material can be forced into the cavity. See Figure 1.2 and Figure 1.3.

Although fluids are usually assumed to be incompressible, molten plastics have to be considered to be more like a gas. The compressibility of plastics can be observed by blocking off the nozzle and attempting to purge the barrel. The ram will jump forward when the pressure is applied, but will spring back when the pressure is released.

1.1.1.3 Compensation Phase

After the pressurization phase, the ram still does not stop completely, continuing to creep forward for some time. Plastics have a very large volumetric change of about 25% from the melt to the solid. This can be seen in a short shot; the difference in volume between the molding and the cavity is due to this volumetric change. See Figure 1.2 and Figure 1.3.

The ram moving forward to compensate for the volumetric change in the part is called the compensation phase. As the volumetric change is 25% and, at the most, only an extra 15% can be injected in the pressurization phase, there must always be some compensation phase.

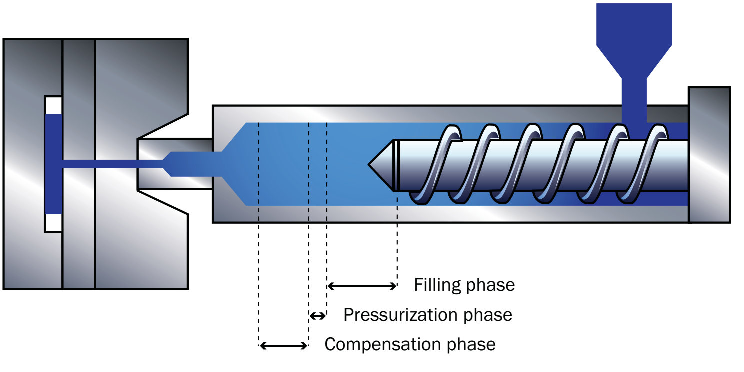


Figure 1.2 Phases of injection molding

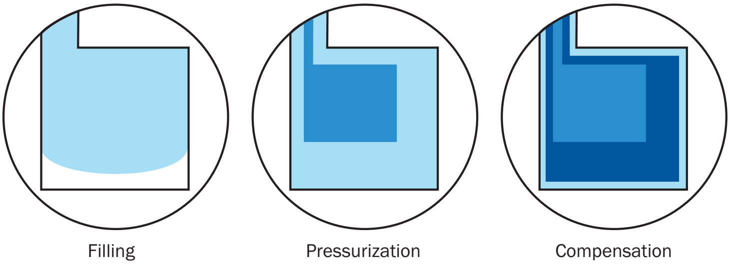


Figure 1.3 Phases of injection molding detail

1.1.2 The Filling Phase

A two-color technique best demonstrates this phase. After emptying the barrel of an injection-molding machine, a small amount of red plastic was charged, followed by green plastic.

Consider the closed mold with the plastic front just starting to flow from the nozzle. The plastic first fills the sprue and runner system, then enters the mold cavity itself, forming a small bubble of molten plastic.

The skin of the plastic in contact with the cool mold freezes rapidly, while the central core remains molten. When additional material is injected, it flows into this central core, displacing the material already there, which then forms a new *flow front*. The flow of this displaced material is a combination of forward flow and outward flow. The outward flow contacts the wall, freezes, and forms the next section of skin while the forward flow forms the new molten core. When more material enters the mold, it flows along a channel lined with these frozen walls of plastic, illustrated in Figure 1.4.

This flow pattern is often called *fountain flow* or *bubble flow* because the flow front is like a bubble being inflated with hot plastic from the center. The frozen layer is formed by the flow front inflating, and so is subject to only a low shear stress and, therefore, has a very low level of molecular orientation. Once it is frozen it cannot be orientated any further, so the frozen layer in the finished part has a low level of orientation.

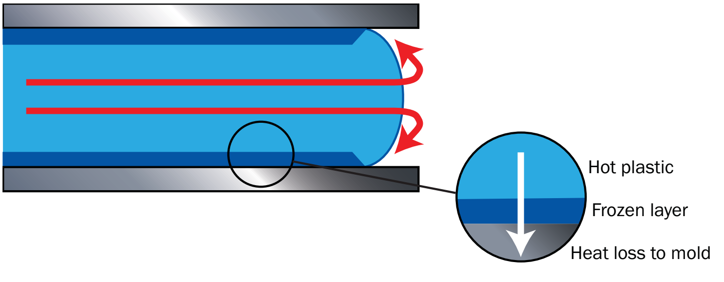


Figure 1.4 Fountain flow and heat transfer

Now, consider what happens upstream. Hot plastic is continuously flowing, bringing new hot material along and generating significant frictional heat. At the same time, heat is being lost through the frozen layer to the cold mold surface.

Initially, the frozen layer is very thin, so heat is lost very rapidly. This results in more plastic freezing and the frozen layer getting thicker, cutting down the heat flow. After a time, the frozen layer will reach a thickness such that the heat lost by conduction is equal to the heat input from plastic flow and frictional heating, i.e., an equilibrium condition is reached (Figure 1.4).

It is interesting to do some calculations on the time taken to reach this state of equilibrium. The actual rate of heat flow is very large in comparison with the small heat content of the plastic in the frozen layer. The result is that equilibrium is reached very quickly, often in a time measured in a few tenths of a second. As the total filling time is measured in seconds, the frozen layer reaches an equilibrium state early in the filling cycle.

It is useful to think about how the thickness of this frozen layer will vary. If the injection rate were slowed, less heat would be generated by friction along the flow path, with less heat input from the flow. The heat loss would be at the same rate, and with less heat input the frozen layer would grow in thickness. If the injection rate were raised, the frozen layer would be thinner (Figure 1.5). Similarly, higher melt and mold temperatures would reduce the thickness of the frozen layer. This can be seen experimentally using the two-color technique.

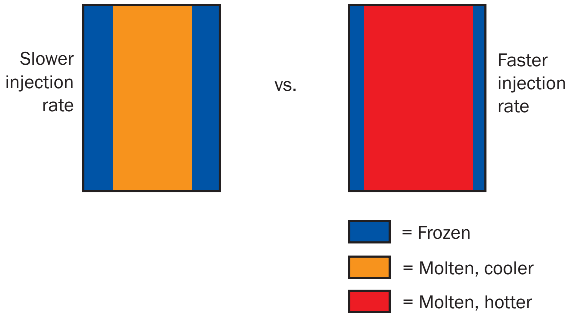


Figure 1.5 Influence of injection rate on frozen layer thickness

1.1.2.1 Flow Shear Stress

It is easy to get confused between the various stress levels and orientation of the polymer. As the plastic flows it is subject to *shear stress*, also called *flow shear stress*. This flow shear stress will orient the material, i.e., cause the molecules to align themselves in the general direction of flow.

The shear stress varies from a maximum at the outside, dropping off to zero at the center.

Shear stress is purely a function of force and area. This must not be confused with shear rate, which is the rate of plastic sliding over the next layer. Shear rate is zero at the outer edge where the plastic is frozen, rises to a maximum just inwards of the frozen layer, then drops toward the center, as shown in Figure 1.6.

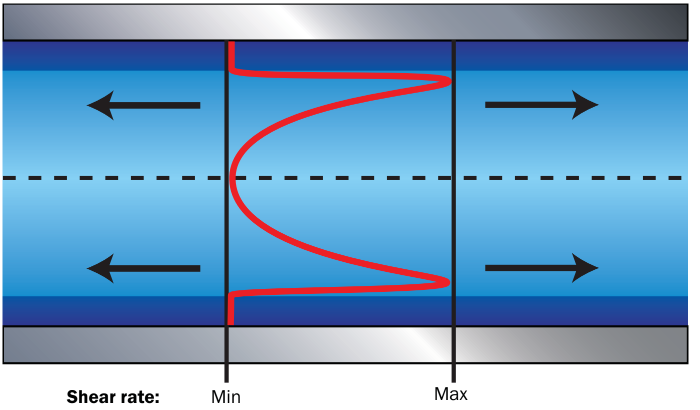


Figure 1.6 Shear rate distribution

If the flow were stopped and the plastic allowed to cool down very slowly, this orientation would have time to relax, giving a very low level of residual orientation. On the other hand, if the material were kept under stress and the plastic snap frozen, most of the orientation would be trapped in the frozen plastic (Figure 1.7).

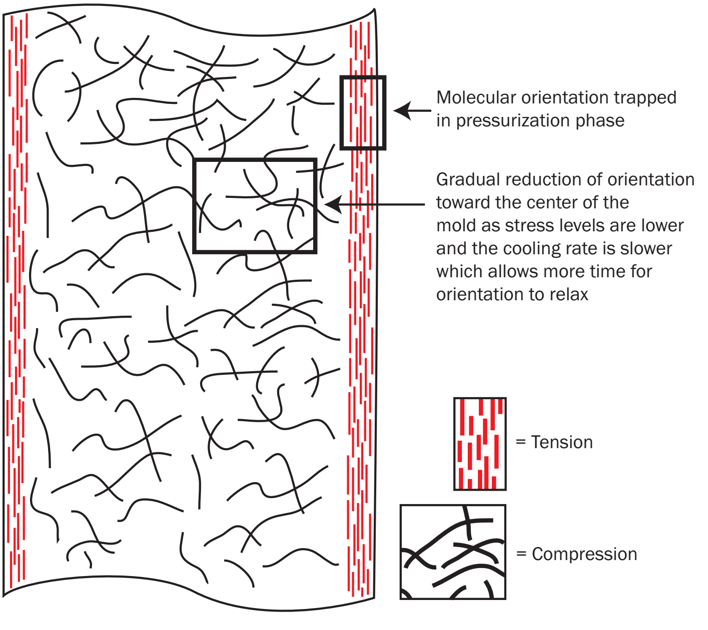


Figure 1.7 Molecular orientation through the thickness of the part

Now consider the orientation from the mold surface toward the center.

The frozen layer itself, formed with very little shear and therefore low orientation, immediately freezes, "setting" the low level of orientation.

The layer of plastic just on the inside of the frozen layer is subject to maximum shear stress and freezes the instant flow stops, trapping almost all the orientation.

This is the orientation pattern: the further toward the center, the more the shear stress drops and the slower the rate of cooling. This allows more time for the level of orientation to relax, so the residual orientation drops rapidly toward the center. Consider how this pattern will affect the residual stress level. Oriented material (normally) will shrink more than nonoriented material. On the inner surface of the original frozen layer, highly oriented material wants to shrink a great deal, but it is prevented from doing so by the less-oriented material. The highly oriented layer ends up being in tension, while the less-oriented material is in compression.

This residual stress pattern is a common cause of part warpage.

> There is a connection—through orientation—between the shear stress during filling (flow stress) and the residual stress in the final molded part. This means shear stress during filling, shown on Moldflow plots, can be used as a design parameter.

1.1.3 The Pressurization Phase

The pressurization phase—from the point of view of flow behavior—is very similar to the filling phase. The flow rate may drop somewhat as the mold builds up to pressure, resulting in an increase in the thickness of the frozen layer.

The main difference of course, is the increase in hydrostatic (isotropic) pressure. We shall see in chapter 2, section 2.4 Effect of Molding Conditions, that hydrostatic pressure in itself does not cause any residual stress.

1.1.4 The Compensation Phase

Compensating flow is unstable. Consider the plate molding again (see Figure 1.1). You would think that plastic flowing uniformly through the thin diaphragm would top up the thick rim. In practice, the plastic during the compensation phase flows in rivers that spread out like a delta, as illustrated in Figure 1.8. This may seem surprising at first, but it can be explained by temperature instability.

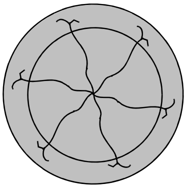


Figure 1.8 River flow

1.1.4.1 Temperature Variation

There is always some variation in melt temperature coming from the barrel of the injection machine. In exceptional cases, up to 40 ºC variation has been measured using a high-speed thermocouple..

1.1.4.2 Natural Instability

However slight the temperature variation, natural instability will amplify it. If, for example, one part of the melt is slightly hotter than the rest, then the plastic flow in that area will be slightly greater, bringing hotter material into the area and maintaining the temperature. If, on the other hand, there is another area that is cooler, the flow will be less, so there will be less heat input, and the plastic will get colder until it eventually freezes off.

However balanced the initial conditions, this natural instability will result in a river-type flow. This is a very important consideration. The first material to freeze off will shrink early in the cycle. By the time the material in the river flows freezes, the bulk of the material will have already frozen off and shrinkage will have occurred. The rivers will shrink relative to the bulk of the molding, and because they are highly orientated, shrinkage will be very high. The result is high-stress tensile members throughout the molding, a common cause of warpage.

1.1.4.3 Optimum Part Quality

Most of the stress in plastic parts occurs during the compensation phase. By controlling flow and minimizing stress, it is possible to design for optimum part quality. This important point is at the heart of the Moldflow philosophy.

1.2 How Do Plastics Flow?

1.2.1 Material Behavior

Molten thermoplastics exhibit viscoelastic behavior, which combines flow characteristics of both viscous liquids and elastic solids. When a viscous liquid flows, the energy that causes the deformation is dissipated and becomes viscous heat. On the other hand, when an elastic solid is deformed, the driving energy is stored. For example, the flow of water is a typical viscous flow, whereas the deformation of a rubber cube falls into the elastic category.

1.2.2 Deformation

In addition to the two types of material flow behavior, there are two types of deformation: simple shear and simple extension (elongation), as shown in Figure 1.9 (a) and (b) below. The flow of molten thermoplastics during injection-molding filling is predominantly shear flow, as shown in Figure 1.9 (c), in which layers of material elements "slide" over each other. The extensional flow, however, becomes significant as the material elements undergo elongation when the melt passes areas of abrupt dimensional change (e.g., a gate region), as shown in Figure 1.9 (d).

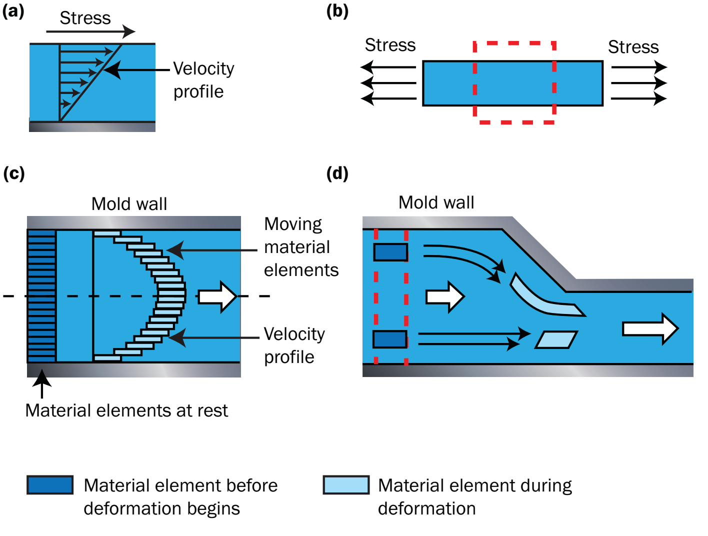


Figure 1.9 (a) Simple shear flow (b) Simple extensional flow (c) Shear flow in cavity filling (d) Extensional flow in cavity filling

1.2.3 Viscoelastic Behavior

In response to an applied stress (force per unit area), molten thermoplastics exhibit viscoelastic behavior, which combines characteristics of an ideal viscous liquid with those of an ideal elastic solid. In other words, under certain conditions, molten thermoplastics behave like a liquid and will continuously deform while shear stress is applied, as shown in Figure 1.10. Upon the removal of the stress, however, the materials behave somewhat like an elastic solid with partial recovery of the deformation, as shown in Figure 1.10 (b) and (c). This viscoelastic behavior stems from the random-coil configuration of polymer molecules in the molten state, which allows the movement and slippage of molecular chains under the influence of an applied load. However, the entanglement of the polymer molecular chains also makes the system behave like an elastic solid upon the application and removal of the external load. Namely, on removal of the stress, chains will tend to return to the equilibrium random-coil state and thus will be a component of stress recovery. The recovery is not instantaneous because of the entanglements still present in the system.

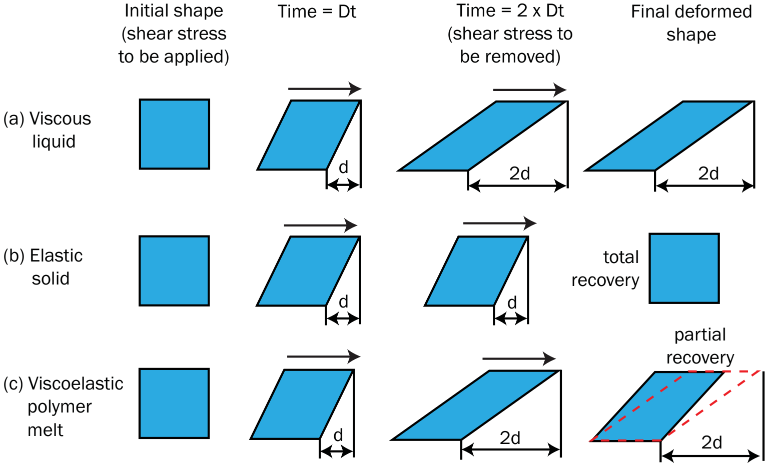


Figure 1.10 (a) Ideal viscous liquid deforms continuously under applied stress (b) Ideal elastic solid deforms immediately upon the application of stress, but fully recovers when the stress is removed (c) Molten thermoplastic deforms continuously under the applied stress (like a viscous liquid), but also recovers partially from the deformation upon removal of the applied stress (like an elastic solid)

1.2.4 Melt Shear Viscosity

1.2.4.1 What Is Shear Viscosity?

Melt shear viscosity is a material's resistance to shear flow. In general, polymer melts are highly viscous because of their long molecular chain structure. The viscosity of a polymer melt ranges from 2 to 3,000 Pa.s (water 10^-1 Pa.s, glass 10^20 Pa.s). Viscosity can be thought of as the thickness of a fluid, or how much it resists flow. Viscosity is expressed as the ratio of shear stress (force per unit area) to the shear rate (rate change of shear strain), as shown in Equation 1.1 and Figure 1.11:

$$\text{viscosity} = \frac{\text{shear stress}}{\text{shear rate}} \tag{1.1}$$

where

$$\text{shear stress} = \frac{\text{force (F)}}{\text{area (A)}} \quad \text{and} \quad \text{shear rate} = \frac{\text{velocity (v)}}{\text{height (h)}}$$

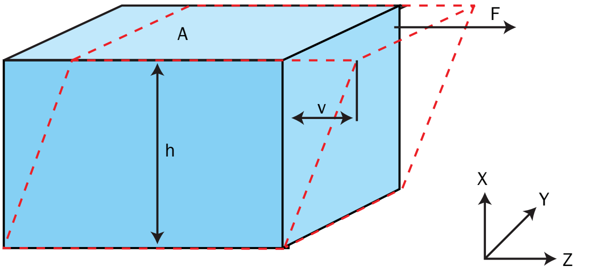


Figure 1.11 The definition of polymer melt viscosity, illustrated by a simple shear flow

1.2.5 Newtonian Fluid vs. Non-Newtonian Fluid

For Newtonian fluids, viscosity is a temperature-dependent constant, regardless of the shear rate. A typical example of Newtonian fluid is water. However, for non-Newtonian fluids, which include most polymer melts, the viscosity varies, not only with temperature but with the shear rate.

1.2.6 Shear-thinning Behavior

When the polymer is deformed, there will be some disentanglement, slippage of chains over each other, and molecular alignment in the direction of the applied stress. As a result of the deformation, the resistance exhibited by polymer to flow decreases, due to the evolution of its microstructure (which tends to align in the flow direction). This is often referred to as shear-thinning behavior, which translates to lower viscosity with a high shear rate. Shear-thinning behavior provides some benefits for processing the polymer melt. For example, if you double the applied pressure to move water in an open ended pipe, the flow rate of the water also doubles because the water does not have shear-thinning behavior. But in a similar situation using a polymer melt, doubling the pressure may increase the melt flow rate from two to 15 times, depending on the material.

1.2.7 Shear Rate Distribution

Having introduced the concept of shear viscosity, let us look at the shear rate distribution in the cavity during injection molding. The faster the adjacent material elements move over each other, the higher the shear rate is. Therefore, for a typical melt flow velocity profile, shown in Figure 1.12 (a), the highest shear rate is just inside the frozen layer. The shear rate is zero at the centerline because there is no relative material element movement due to flow symmetry, as shown in Figure 1.12 (b). Shear rate is an important flow parameter because it influences the melt viscosity and the amount of shear (viscous) heating. The typical shear rate experienced by the polymer in the cavity is between 100 and 10,000 1/seconds. The feed system can see shear rates in excess of 100,000 1/seconds.

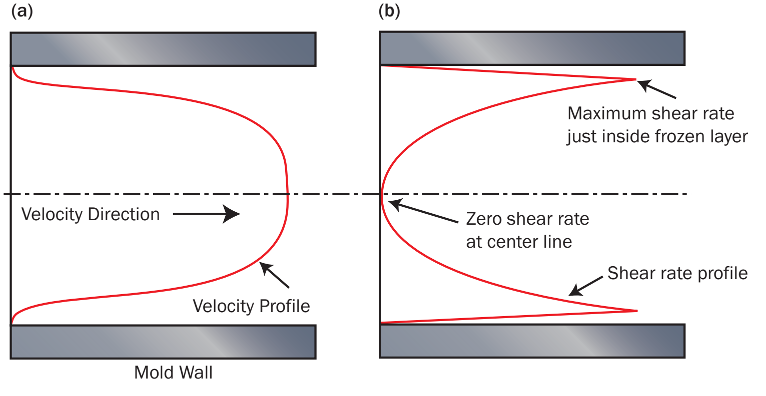


Figure 1.12 (a) A typical velocity profile (b) The corresponding shear rate distribution in injection molding filling

1.2.7.1 Effects of Temperature and Pressure

Since the mobility of polymer molecular chains decreases with decreasing temperature, the flow resistance of polymer melt also greatly depends on the temperature. As shown in Figure 1.13, the melt viscosity decreases with increasing shear rate and temperature because of the disentanglement and alignment of the molecules and enhanced mobility of polymer molecules, respectively. In addition, the melt viscosity also depends on the pressure. The higher the pressure, the more viscous the melt becomes.

1.2.8 Pressure-driven Flow

The flow of molten thermoplastics during the filling phase is driven by pressure that overcomes the melt's resistance to flow. Molten thermoplastics flow from high-pressure to low-pressure areas, analogous to water flowing from higher elevations to lower elevations. During the filling phase, high pressure builds up at the injection nozzle to overcome the flow resistance of the polymer melt. The pressure gradually decreases along the flow length toward the polymer melt front, where the pressure reaches the atmospheric pressure, if the cavity is vented properly (see Figure 1.14).

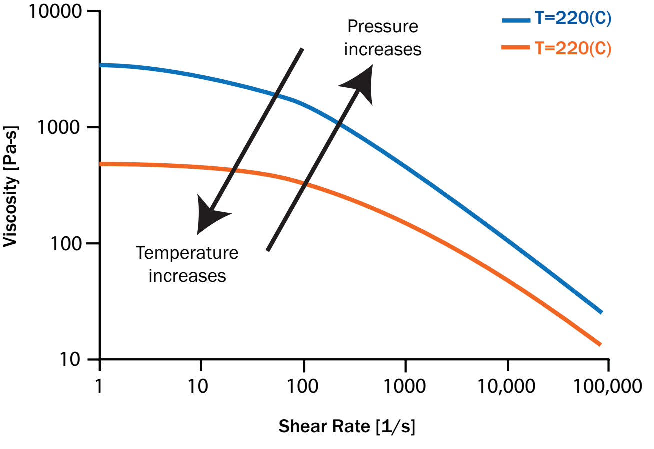


Figure 1.13 The viscosity of polymer melt depends on the shear rate, pressure, and temperature

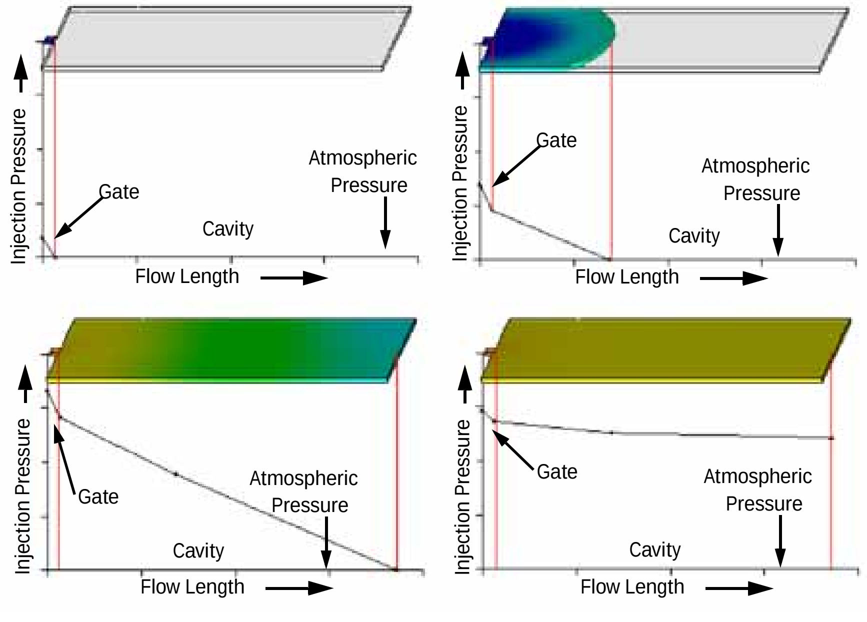


Figure 1.14 Evolution of pressure distribution within the cavity during the filling and early packing stages

1.2.9 Pressure Gradient and Injection Times

The filling phase should be controlled by injection time, i.e., velocity. The part should be filled so that the pressure gradient (pressure drop per unit flow length) is constant during the filling. To maintain a constant pressure gradient, the pressure at the machine nozzle continues to increase as the flow front progresses through the part. Figure 1.15 shows pressure traces for three different injection times. The pressure gradient (the slope of the line) is different for each fill time; a faster fill time results in a steeper pressure gradient. However, for each fill time, the rate of change in pressure per unit of time is nearly uniform.

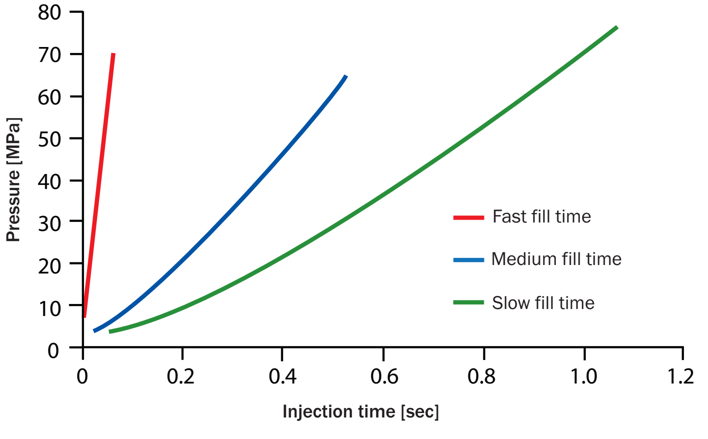


Figure 1.15 The relationship of injection time to pressure gradient

1.2.10 Melt Flow Length

During injection molding, the distance that the material can flow, with certain processing conditions and wall thickness, depends on the thermal properties and shear properties of the material. This behavior can be characterized by the melt flow length, as illustrated in Figure 1.16.

1.2.11 Injection Pressure vs. Fill Time

For injection molding, if the injection pressure required to fill the cavity is plotted against the time to fill the cavity, a U-shaped curve results, with the minimum value of the required injection pressure occurring at an intermediate fill time, as illustrated in Figure 1.17. The curve is U-shaped because a short fill time involves a high flow rate, thus requiring a higher injection pressure to fill the mold. With a long fill time, there is less shear heat being generated. This will increase the viscosity of the polymer and lower the polymer temperature in the part, therefore increasing the pressure required to fill. The scale of the U-shaped curve of injection pressure versus fill time depends on the material used, the flow length, and wall thickness of the part. The optimum time range is not only based on pressure, but also on the polymer's temperature variation and the shear stress developed during the filling phase.

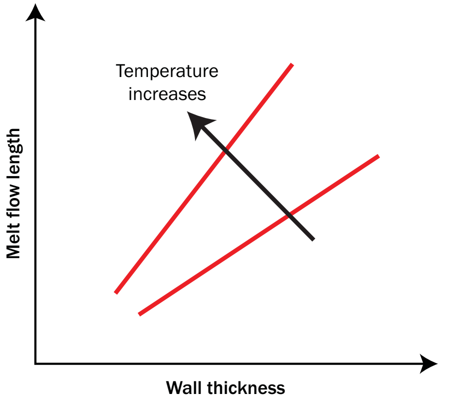


Figure 1.16 The melt flow length depends on the part thickness and temperature

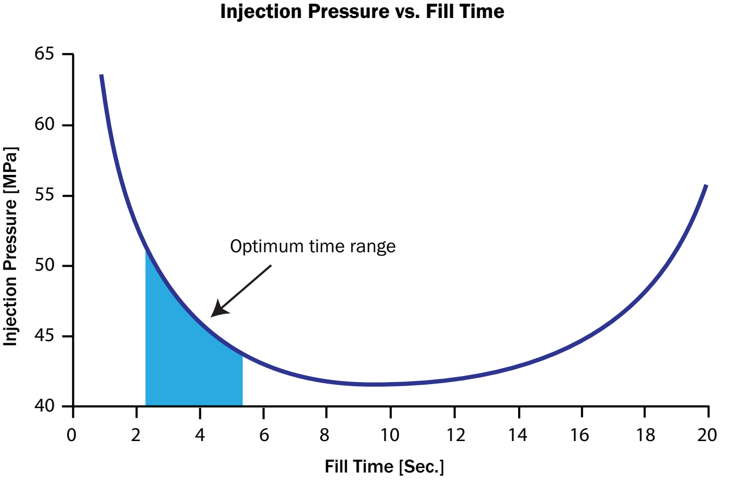


Figure 1.17 U-shaped curve of injection pressure vs. fill time

1.2.12 Flow Instability

The dynamics of cavity filling may sometimes become quite complicated because of the interaction of the melt velocity (or, equivalently, the shear rate), the melt viscosity, and the melt temperature. Recall that the melt viscosity decreases with increasing shear rate and temperature. It is possible that high shear rate and shear heating resulting from a higher melt velocity will drive the viscosity down, so that the flow velocity actually increases. This will create a greater shear rate and temperature rise, and is an inherent instability of highly shear-sensitive materials.

2 Molding Conditions and Injection Pressure

- Injection pressure overview
- Factors that influence injection pressure requirements
- Equations
- Effect of molding conditions
- Using Moldflow to determine optimum processing conditions

2.1 Injection-pressure Overview

Pressure, pushing the polymer to fill and pack the mold cavity, is the driving force that overcomes the resistance of polymer melt. If you place a number of pressure sensors along the flow path of the polymer melt, the pressure distribution in the polymer melt can be obtained, as schematically illustrated in Figure 2.1.

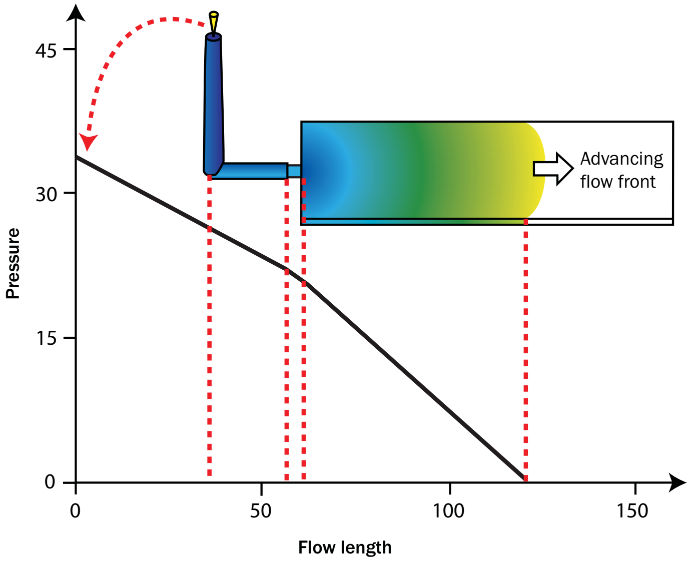


Figure 2.1 Pressure decreases along the delivery system and the cavity.

2.1.1 Pressure Drives the Flow Front

The polymer flow front travels from areas of high pressure to areas of low pressure, analogous to water flowing from higher elevations to lower elevations. During the injection stage, high pressure builds up at the injection nozzle to overcome the flow resistance of the polymer melt. The pressure decreases along the flow length toward the polymer flow front, where the pressure reaches the atmospheric pressure if the cavity is vented. Broadly speaking, the pressure drop increases with the flow resistance of the melt, which, in turn, is a function of the geometry and melt viscosity. The polymer's viscosity is often defined with a melt flow index. However, this is not a good measure of the material's behavior during the filling phase. As the flow length increases, the polymer entrance pressure increases to maintain a desirable injection flow rate.

2.2 Factors Influencing Injection-pressure Requirements

The following diagrams illustrate the design and processing factors that influence injection pressure.

Table 2.1: Factors influencing injection pressure

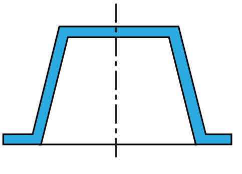
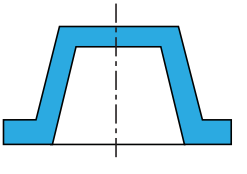
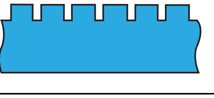
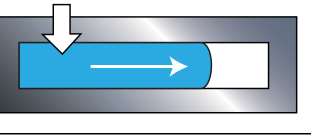
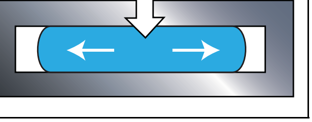
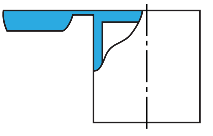
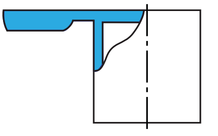
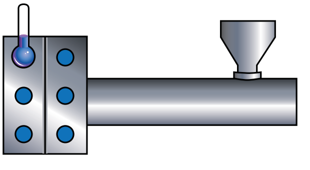
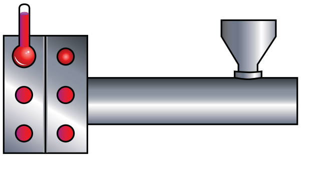
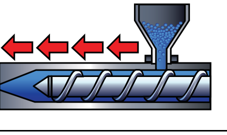
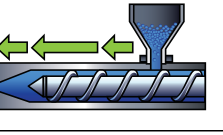
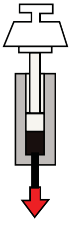
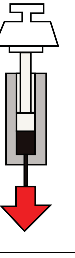
Factor	Variable	Higher injection pressure required	Lower injection pressure required
Part design	Part thickness	Thin part 	Thick part 
	Part surface area	More wall cooling and drag force 	Less wall cooling and drag force 
	Flow length	Long flow length 	Short flow length 

Table 2.1: Factors influencing injection pressure

Factor	Variable	Higher injection pressure required	Lower injection pressure required
Feed system design	Gate size	Restrictive gate 	Generous gate 
	Runner diameter	Runner diameter too small or too large 	Runner diameter optimized 
Processing conditions	Mold temperatire	Colder coolant temperature 	Hotter coolant temperature 
	Melt temperature	Colder melt temperature 	Hotter melt temperature 
	Ram speed (injection time)	Improper ram speed 	Optimized ram speed 
Material selection	Melt flow index	Low index material 	High index material 

2.3 Equations

Based on a simplification of classic fluid mechanics theory, the injection pressure required to fill the delivery system (the sprue, runner, and gate) and cavities can be correlated with several relevant material, design, and processing parameters. In the following equations, *P* is the injection pressure and *n* is a material constant (the power-law coefficient), which typically ranges from 0.15 to 0.36 (with 0.3 being a good approximation) for a variety of polymer melts. Figure 2.2 shows injection pressure as a function of several of these parameters.

2.3.1 Circular Channel Flow

Circular channel flow describes the melt flow in the sprue, runner, and cylindrical gates:

$$P \propto \frac{(\text{melt viscosity})(\text{flow length})(\text{volumetric flow rate})^{n}}{(\text{channel radius})^{3n+1}} \tag{2.1}$$

2.3.2 Strip Channel Flow

Strip channel flow describes the melt flow in a thin cavity:

$$P \propto \frac{(\text{melt viscosity})(\text{flow length})(\text{volumetric flow rate})^{n}}{(\text{channel width})(\text{channel thickness})^{2n+1}} \tag{2.2}$$

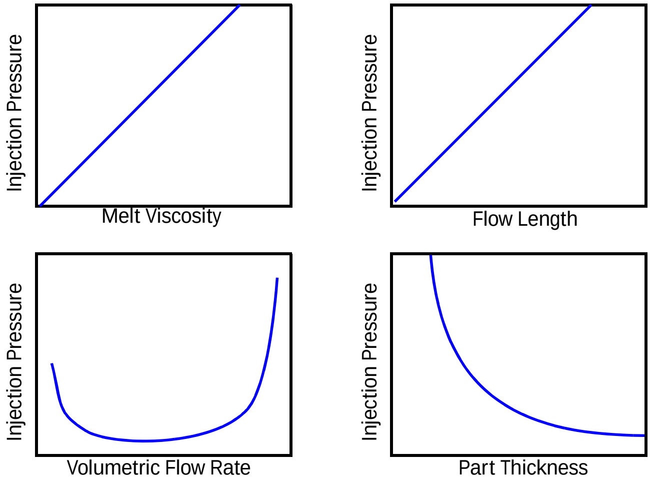


Figure 2.2 Injection pressure as a function of melt viscosity, flow length, volumetric flow rate, and part thickness

2.4 Effect of Molding Conditions

Consider the effect of the key molding settings of mold temperature, melt temperature, and fill time on part quality.

2.4.1 Part Quality

First, part quality must be defined. The main aims must be minimal residual stress level and the avoidance of both warpage and sink marks. Residual stress levels can be checked in one of two ways: contour map or reversion test.

If the part is transparent, it can be viewed through polarized sheets and the stress level read like a contour map as shown in Figure 2.3.

If the part is opaque, the reversion test must be used. A family of circles is drawn on the part; then the part placed in an oven at a specified temperature for a given time. The major and minor axes of the ellipse formed by the deformation of the drawn circles are measured, and the ratio gives some measure of the residual stress or orientation level, as shown in Figure 2.4.

Figure 2.3 Transparent tape dispenser shown through polarized sheets; dense color changes indicate high residual stress

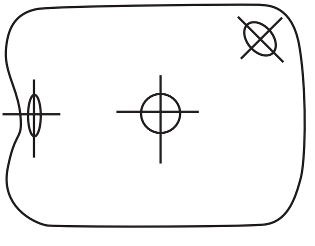


Figure 2.4 Reversion test to determine orientation level

2.4.2 Melt Temperature

Figure 2.5 indicates the effect of melt temperature on part weight and stress. At low temperatures, both pressure to fill the mold and shear stress levels within the cavity itself are initially high. As the melt temperature increases, the curve flattens out, producing a smaller reduction in shear stress for a given increase in melt temperature. Of course, the rate of material degradation increases as melt temperature is raised, so too high temperatures may result in lower quality parts.

Even though raising the melt temperature means the part can be packed with more pressure, part weight is reduced because there will be a large increase in volumetric shrinkage that can't be compensated for by the change in pressure requirements. Sink marks will increase, as indicated by the reduction in part weight.

✎ Part weight varies with temperature and is a useful measure of sink marks.

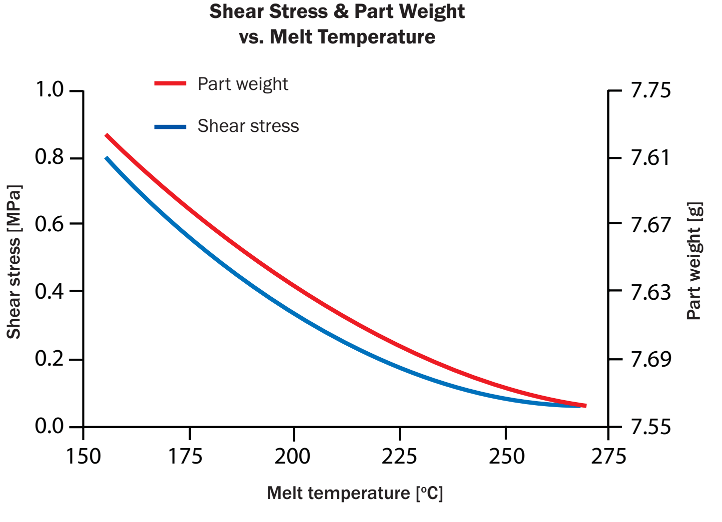


Figure 2.5 Shear stress and part weight vs. melt temperature

2.4.3 Mold Temperature

Increasing mold temperature has a similar effect to melt temperature, except that the effect on pressures and stress levels are less marked until very close to the transition temperature. The effect on cooling time can be much larger than an equivalent change in melt temperature. Often the most important benefit from raising mold temperature is that it allows a slower injection rate, without the plastic getting too cold.

2.4.4 Fill Time

Figure 2.6 shows the effect of varying injection times on pressure to fill and flow front temperature. Again there are conflicting requirements. At very short injection times there are very high shear rates, so the pressure required to fill the cavity is high. Increasing the injection time will give a lower shear rate but more heat will be lost, so the flow front temperature will get very cold increasing in viscosity. This combination of lowering shear rates and temperatures gives this classic U-shaped graph.

Short fill times require relatively high pressures simply because the flow rate is so high. Long injection times require relatively high pressures because the flow front temperature at the end of flow is so cold. Somewhere between these extremes is an injection time that gives an acceptable fill pressure.

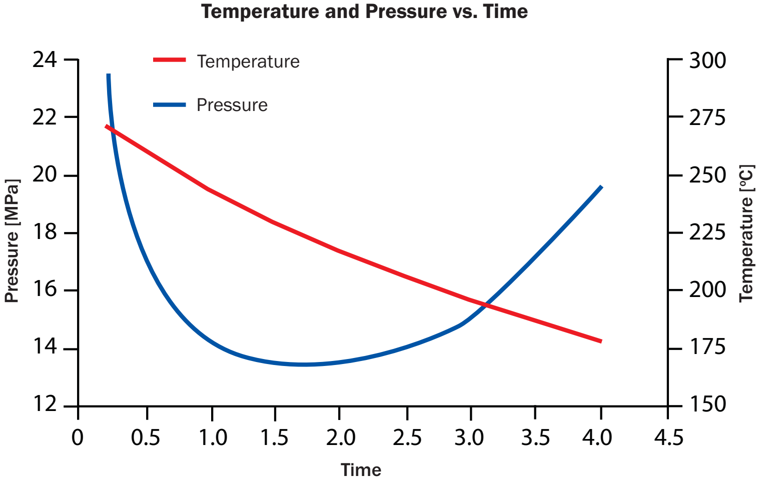


Figure 2.6 Pressure and temperature vs. time

2.4.5 Shear Stress Variation

The shear stress distribution depends on whether it is at the beginning or end of flow. At the beginning of flow, there is no time for heat loss, so the stress is determined largely by shear rate. This means that as the flow is slowed down, the stress levels get consistently lower as shown in Figure 2.7. At the end of flow there is again the conflict between high shear rates at short injection times and low temperatures at long times. In many cases this will give a U-shaped curve, but in other cases a continuous rise in stress level can be seen as fill time is increased.

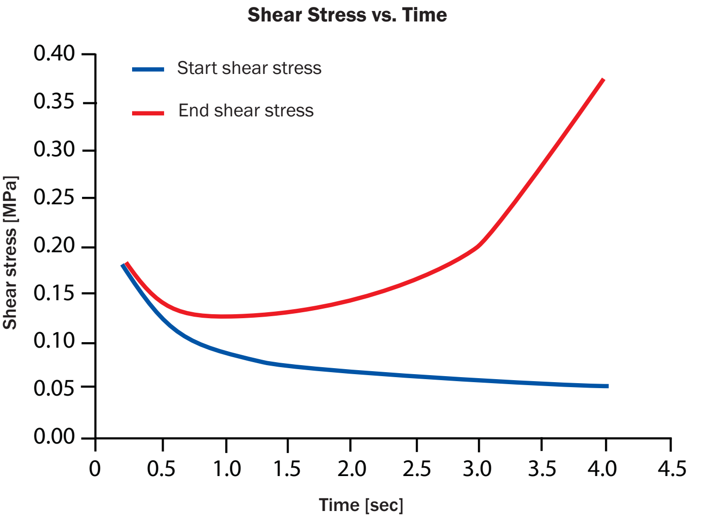


Figure 2.7 Shear stress vs. time

2.4.6 Packing Pressure and Time

Consider the effect of packing time. The filling phase was kept the same throughout the experiment; only the packing pressure and time were varied. A whole family of parts, all the same weight but made with different combinations of packing time and pressure, were produced (i.e., some parts were made with a high packing pressure and held on for a small period of time). See Figure 2.8. Other parts of the same weight were produced but with a low packing pressure and held on for a long period of time. After examining the parts for stress levels, it was found that parts made with the high packing pressure held on for a shorter period of time generally had a lower stress level than parts made with a lower pressure and held on for longer packing time.

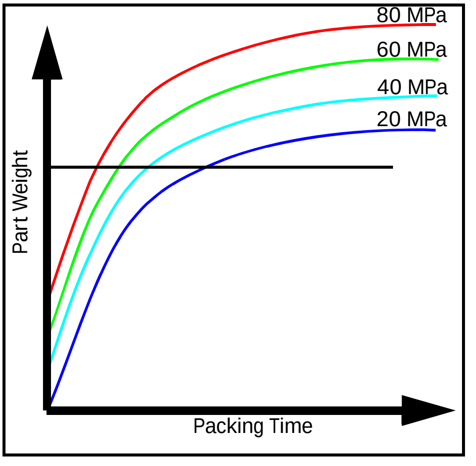


Figure 2.8 Pack pressure and pack time vs. part weight

2.4.7 Summary

Hydrostatic pressure (i.e. all-round pressure) does not cause stress. If a piece of plastic was put into a pressure vessel and pressure applied, the pressure by itself is not going to cause stress or failure (Figure 2.9).

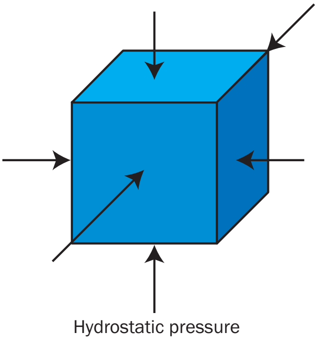


Figure 2.9 Hydrostatic pressure on a polymer

The cause of stress in plastic parts is the combination of the plastic material flowing and freezing at the same time. This combination of conditions occurs during the holding or compensating phase (Figure 2.10).

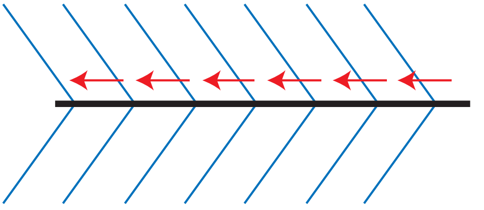


Figure 2.10 Residual stress caused by a combination of flow and freezing at the same time

2.4.8 Back Flow

Back flow is a special situation that can inadvertently occur. Using the same experiment as for the investigation into the effect of packing time as described previously, in certain cases it is possible to reverse the flow. If the mold is brought up to a very high pressure, an extra 15% of material will be forced into the mold due to pressurization. If the packing pressure is then dropped off, the plastic will flow back out of the mold and down the runner. This reverse flow has the same effect as forward flow. Stress is caused by the combination of flow and freeze whether the flow is in or out of the mold. The ideal molding situation is to bring the mold up to pressure, hold the mold under pressure for the minimum time to reduce sink marks to an acceptable level, then have the runner system freeze off so no plastic can flow into or out of the mold.

2.5 Using Moldflow to Determine Optimum Molding Conditions

Moldflow has a molding window analysis that runs very quickly and can be used to evaluate many things including:

- Optimum molding conditions
- Size of molding window
- Material selection
- Pressure required to fill a part
- Gate locations
- Wall thickness

The molding window analysis is a preliminary analysis, but it can answer significant questions and narrow down the focus of detailed analysis very quickly. In the example below, two materials will be evaluated.

2.5.1 Part

A grill, shown in Figure 2.11, will be molded with the five gates indicated by the cones on the bottom edge of the part. The five gates are used to produce a unidirectional and balanced fill. These concepts are discussed in chapter 1.

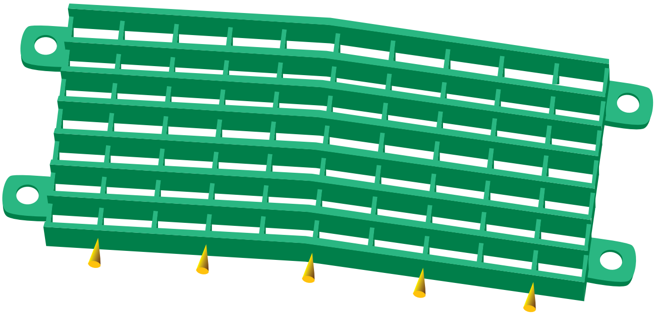


Figure 2.11 Grill with five gate locations

The grill will be molded from ABS. The material selection has been narrowed down to two materials from the same supplier. A molding window analysis will be run to compare both materials.

2.5.2 Molding Window Size

The molding window analysis was run so both materials were evaluated at the same ranges for:

- Mold temperature
  - Range 40–80°C (104–176°F)
- Melt temperature
  - Range 200–280°C (392–536°F)
- Injection time
  - 0.3–10.0 seconds

2.5.2.1 Zone Plot

Figure 2.12 shows the zone plot for both ABS materials. The x-axis is melt temperature and the y-axis is injection time. The zone plot can have three areas:

- Red, not feasible
  - The pressure to fill is over 80% of the machine capacity

- Yellow, feasible
  - The pressure to fill is under 50% of the machine capacity. Pressure or another parameter may be outside the limit
- Green, preferred. All parameters are within acceptable limits, including:
  - Pressure, less than 50% of machine capacity
  - Shear stress, less than the shear stress limit for the material
  - Shear rate, less than the shear rate limit for the material
  - Flow front temperature, within +0 to -20°C (-36°F) from the melt temperature
  - Clamp force, less than 80% of the machine's capacity

For both materials, the size of the preferred window is not huge, but it is large enough so both materials would be easily moldable.

The parameters that define the zone areas can be changed. The pressure limit of 50% of the machine capacity is a reasonable number to use for two reasons. The analysis is done with only gate locations defined on the part. There is no feed system. Having the pressure limit lower ensures there will be pressure to fill the runner. Also, even considering the pressure drop in the runner, the entire tool should not take more than about 75% of the machine capacity if at all possible.

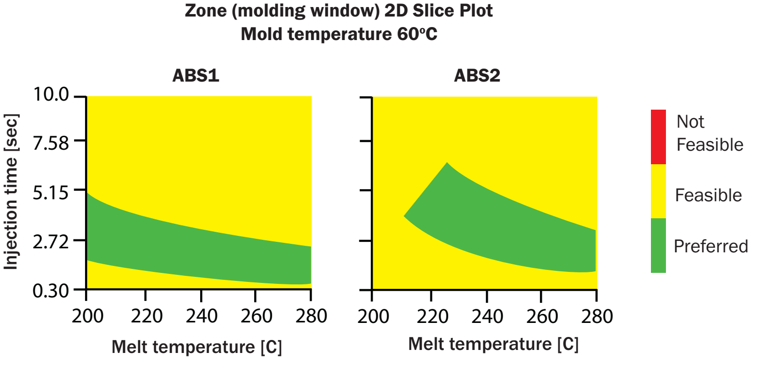


Figure 2.12 Molding window zone plots comparing two grades of ABS

2.5.3 Injection Pressure

The zone plot tells you that the injection pressure is less than 50% of the machine capacity in the preferred area, but you have no idea how much less. Flow front temperature is the main factor that determines the size of the molding window in most cases.

Figure 2.13 shows the pressure to fill the grill at many different injection times from 0.3 seconds to 10.0 seconds at a mid-range mold temperature of 40ºC (104ºF) and melt temperature of 240ºC (464ºF). The graph shows that there are distinct differences between the two materials. ABS2 requires about double the injection pressure as ABS1, but the maximum pressure for ABS2 is still well under half the machine capacity, assuming a typical machine capacity of 140 MPa (20,300 psi).

Both materials can be used to mold the part, but if there are no special reasons to pick ABS2, ABS1 is the best choice because it requires the lower pressure to fill.

2.5.4 Flow Front Temperature

The minimum flow front temperature in the part during the fill is typically the limiting factor in choosing an injection time given a mold and melt temperature. The mold and melt temperatures evaluated are the same as the pressure. Figure 2.14 shows the minimum flow front temperature for both ABS materials over a range of injection times. Even though the melt temperature is the same, the curves are distinctly different because the thermal properties are different for the two materials. The vertical lines represent injection time range that has an acceptable range of flow front temperatures.

ABS2 has nearly a three-second range while ABS1 has a two-second range. Both ranges are good, however, when the part is in production, having a wider time range that can make acceptable parts is always preferred. Based on this information, ABS2 is the best choice.

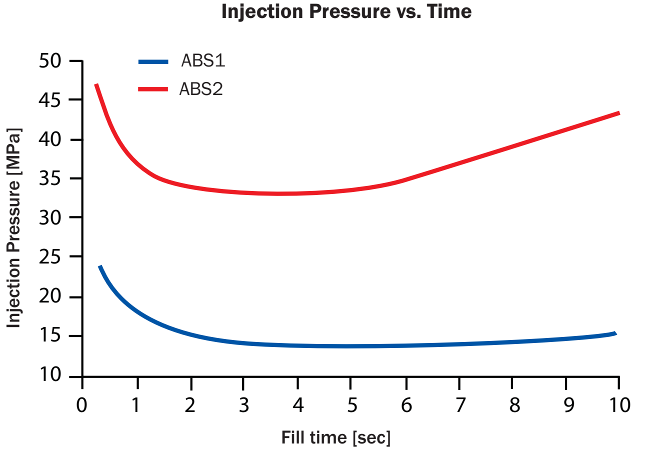


Figure 2.13 Fill times vs. injection pressure to fill comparing two grades of ABS

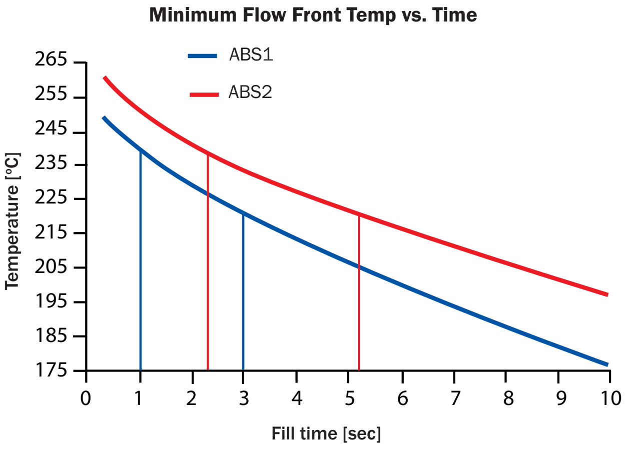


Figure 2.14 Minimum flow front temperature vs. time comparing two grades of ABS; the vertical lines represent the molding window based on the flow front temperature

2.5.5 Cooling Time

The time the materials take to cool to the point of ejection can be a critical factor in determining the profitability of a part. Mold temperatures were evaluated within the materials recommended range. Figure 2.15 shows the cooling times at various mold temperatures for both materials. Due to the thermal property differences of the materials, the cooling time of ABS2 is nearly 12 seconds or 40% faster to cool at the same recommended mold temperature than ABS1. Over the life of the tool, this is a tremendous cost savings.

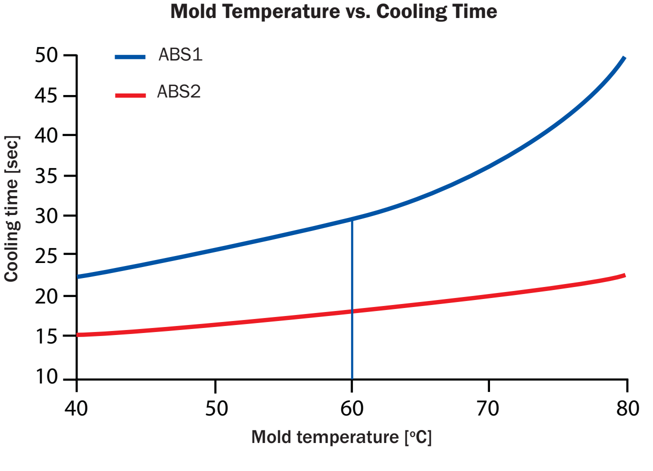


Figure 2.15 The mold temperature vs. cooling time comparing two grades of ABS; the vertical line represents the picked mold temperature

2.5.6 Summary

From the initial choice of the two ABS materials, an initial assumption would be that the easier-to-flow material (in this case ABS1) based on a melt flow index (or some other measure) would be the best material. However, a quick molding window analysis (taking less than 30 minutes for both parts, including interpretation of results) clearly shows that ABS2, which takes more pressure to fill, is the best choice based on the cooling time and size of the molding window. A more detailed analysis can validate the gate locations, size the gates and runners, design the cooling system, and ensure the part will not warp too much.

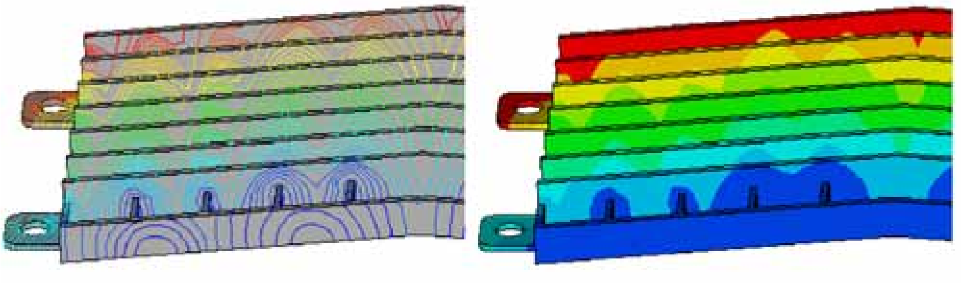
3 Filling Pattern

- Filling pattern overview
- Flow in complex molds
- Flow front area and flow front velocity
- Using Moldflow to determine the filling pattern

3.1 Filling Pattern Overview

3.1.1 What Is the Filling Pattern?

The filling pattern is the transient progression of the polymer flow front within the feed system and mold cavities. It plays an important role in determining the quality of the part and is one of the most important results from a filling analysis. Figure 3.1 illustrates two examples of a filling pattern. The image on the left shows the filling pattern displayed as contour lines. The distance between the lines represents an equal amount of time. The image on the right shows the filling pattern as a shaded image with the colors in bands. These images represent just two ways filling patterns may be represented.



Filling pattern as contour lines   Filling pattern as a shaded image

Figure 3.1 Filling patterns

3.2 Flow in Complex Molds

The filling pattern is a key plot used to investigate many problems. Many of these problems only occur in part designs that have more complex shapes.

3.2.1 Overpack

Overpack is one of the most common causes of warping. Plastics are highly compressible materials. In single, multicavity, or family molds the main cause of overpack and, hence, warping is unbalanced flow. The flow front will always fill the easiest flow path first. Thus, in a single cavity mold, where one area is much easier to fill than another, the plastic will fill the easy area first and continue to pack this area while material reaches the other areas. Figure 3.2 shows a grill with four gates along the bottom edge. There is no center vertical rib, so once the horizontal ribs fill to the center, the area overpacks. This figure shows the filling pattern as a contour line and the pressure distribution as the shaded background and weld lines.

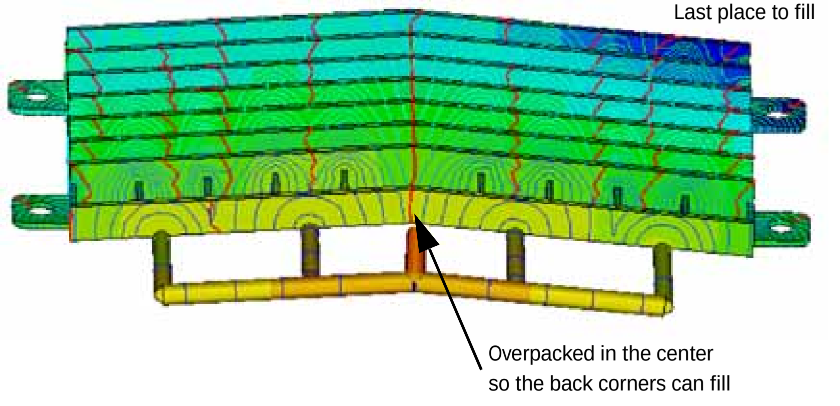


Figure 3.2 Overpacking on a grill

The process whereby overpack causes so much stress can be explained by considering a combination of effects. At the instant that the mold is filled, there will be the traditional zone just inside the frozen layer of highly orientated material. This is unavoidable. While the rest of the part is still filling, plastic in the overpacked area will continue to flow at a gradually decreasing rate, steadily increasing the thickness of the frozen layer. As each new layer of frozen material is formed, the area will have the combination of simultaneous flow and freezing, resulting in the varied orientation of the whole cross section, which sets up its own local stress field. Other areas of lower pack will have lower levels of both orientation and shrinkage that will set up variations in global shrinkage, leading to a global variation in residual stress resulting in warpage.

3.2.2 Racetrack Effect

Figure 3.3 shows an example of the *racetrack effect*. The part consists of a thin top, nominal wall on the sides, and a heavy rim. The part is gated in the rim. Racetracking occurs because the polymer will take the path of least resistance, therefore favoring the heavy rim. The flow front will "race" around the heavy rim. This will often cause serious problems with filling, including air traps and weld lines. With an understanding of the basic principles involved, it is possible to completely control which flow path fills first. Two factors are at work: fluid flow and heat transfer. The final pressure is a combination of these two factors.

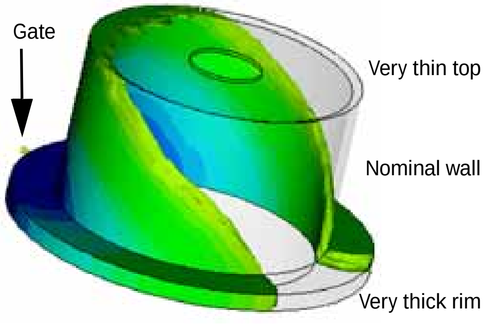


Figure 3.3 Racetracking on a cover

3.2.3 Varying Injection Rate

If injection is very slow, there will be a high heat loss, causing the frozen layer to inhibit the flow in the thin section of Figure 3.4 (a). This is "heat transfer dominated flow." The flow will still be relatively fast in the thick section, and may create an air entrapment problem in many parts. Increasing the fill rate as shown in Figure 3.4 (b), will reduce the thickness of the frozen layer and will preferentially increase the flow in the thin section, relative to the thick section. The thickness of the frozen layer can also be reduced by increasing the melt and mold temperatures.

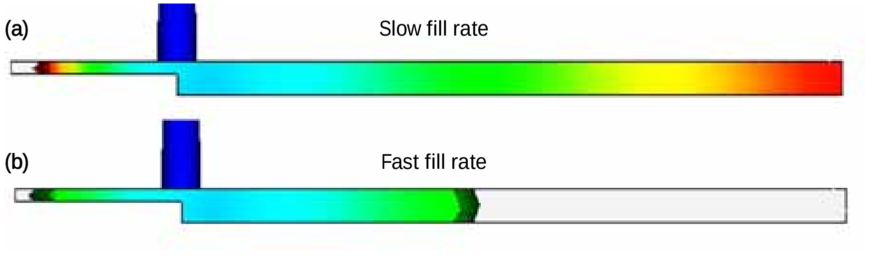


Figure 3.4 Varying the injection rate

3.2.4 Underflow Effect

Another flow problem is the *underflow effect*. Notice the filling pattern in Figure 3.5. The flow from each side gate meets the center flow, forming a weld line, then stops, and reverses direction. Figure 3.6 shows a close-up of one of the weld lines. When the flow stops, the frozen layer will increase in thickness, then remelt due to frictional heat, as the flow starts in the other direction. This flow reversal produces poor part quality, both from surface appearance and structural viewpoints. The arrows shown in Figure 3.6 are velocity vectors, which should be always perpendicular to the fill time contours. When they are not, it indicates underflow has taken place.

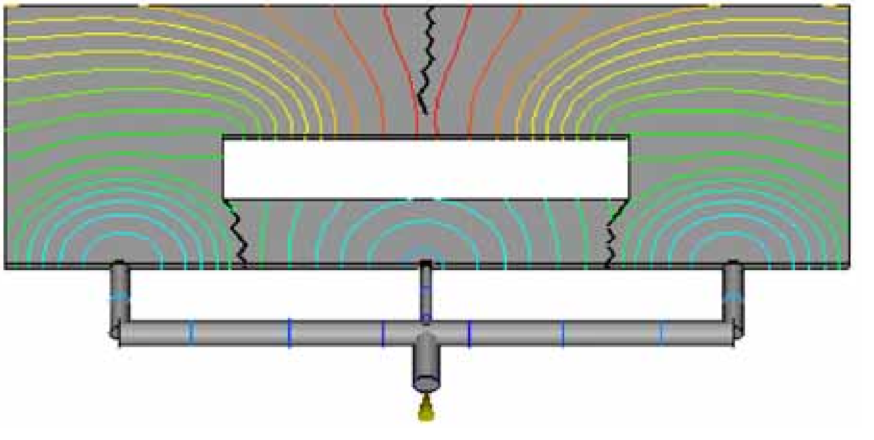


Figure 3.5 Underflow due to gating locations

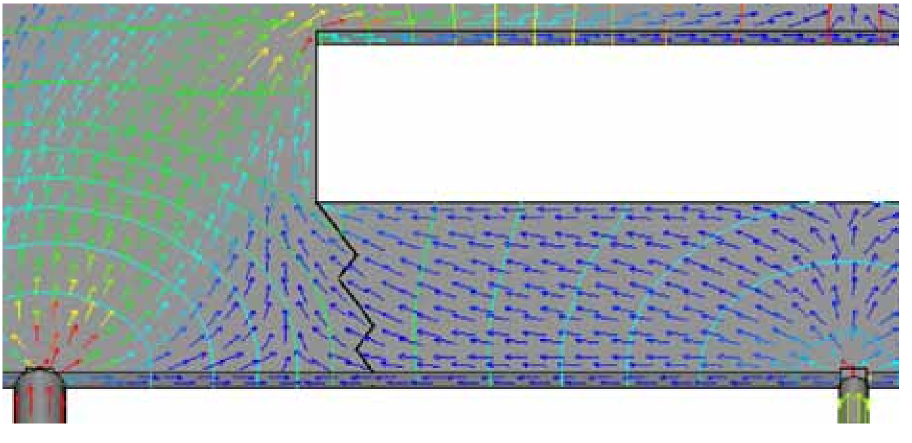


Figure 3.6 Underflow, contour lines and velocity arrows should be perpendicular

3.2.5 Hesitation Effect

To understand the *hesitation effect*, see the example in Figure 3.7. This is the same part as shown in Figure 3.3, but in Figure 3.7 shows the fill pattern just before the part filled. The plastic first enters the gate from the upper left corner. The flow front reaches the thin area about halfway during the fill during the fill. At this time there is insufficient pressure to fill this thin area because the plastic has an alternate route along the thick section. (The alternate route is the racetrack effect discussed earlier.) Plastic that has just entered the thin section slows down considerably and loses heat until the rest of the mold is filled. As injection pressure builds, the flow front in this thin area may start to move faster. If the hesitation is severe enough, the flow front will freeze off, preventing the part from filling completely.

The thin area furthest from the gate fills easily because there are no thicker areas left to fill so the pressure builds until it forces material through the thinner area. The flow path that fills out the part is shown in Figure 3.7. This path is symmetrical on both sides of the part.

> The racetrack and hesitation effects are opposite of each other. Racetracking will occur if a wall section is heavier than nominal, and hesitation will occur in thinner areas. The severity of either of these effects depends on the change in wall thickness, the material, and molding conditions. The best solution to fix either problem is to reduce the wall thickness variation.

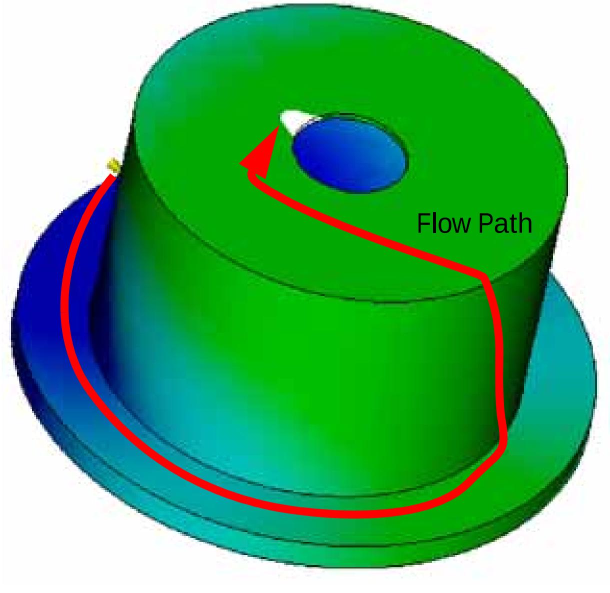


Figure 3.7 Hesitation effect

3.2.6 Weld Lines

Weld lines are formed when two flow fronts meet head on, as shown in Figure 3.8. In multigated parts weld lines are unavoidable. As well as being visually unacceptable, they give the part areas of local weakness because they act as stress concentrators. Although the weld lines cannot be eliminated, a flow analysis can show their location. The mold can then be redesigned to position weld lines in the least sensitive area, considering both structural and aesthetic demands. The pressures and temperatures of each of the converging flow fronts indicate the quality and position of the weld lines. The gate(s) location may be adjusted or relocated, or the runner size can be adjusted

3.2.7 Meld Lines

A meld line is similar to a weld line, except the flow fronts move in parallel rather than meet head on (Figure 3.8). In many cases weld lines turn into meld lines. Initially, when the two flow fronts meet in Figure 3.8, a weld line is formed. Then the line turns into a meld line. Both weld lines and meld lines should be avoided if possible by minimizing the number of gates and placed in the least sensitive areas.

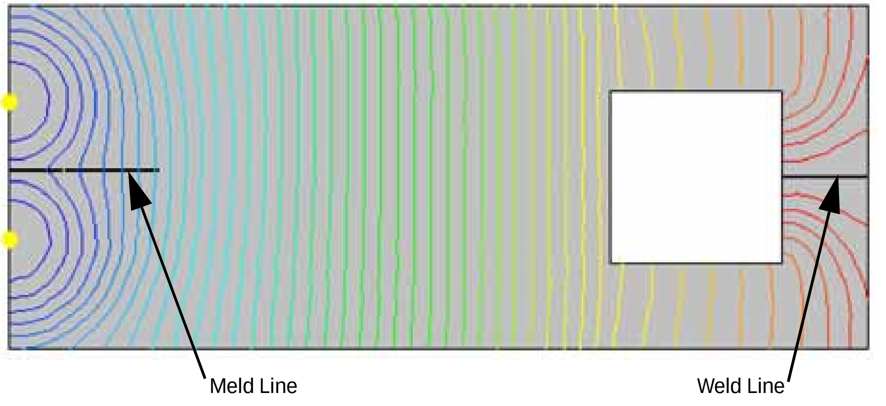


Figure 3.8 Weld and meld lines

3.2.8 Sink Marks

Sink marks are an indentation on the surface where there is a significant local change in wall section (e.g., ribs caused by thermal contraction). An example is shown in Figure 3.9.

It is impossible to eliminate some sink marks by using a high packing pressure, yet using a lower packing pressure produces a part with an acceptable sink mark. This defies common sense, yet the explanation brings up certain important concepts. Because the volumetric change of plastic from the melt to the solid is about 25% and the compressibility of plastics at an injection-molding pressure is only about 15%, it is impossible to pack out a mold and prevent sink marks in the pressurization phase only. Some compensating flow is necessary to eliminate the sink marks by forcing plastic through the thin section to pack out the thick boss. Plastic flow is a combination of viscous flow and heat transfer. Remember the U-shaped graph in chapter 2, Figure 2.6. With a very slow flow rate, the pressure drop will be high because of the high heat loss. In the extreme, the plastic could freeze off. With a high holding pressure, there would be a high flow in the pressurization phase and a low flow in the compensating phase. This low compensating phase flow means that the thin section would not remain molten for a long enough period of time for the boss to be adequately packed out. This way of thinking about flow as a combination of fluid flow and heat transfer effects can aid in understanding many confusing situations.

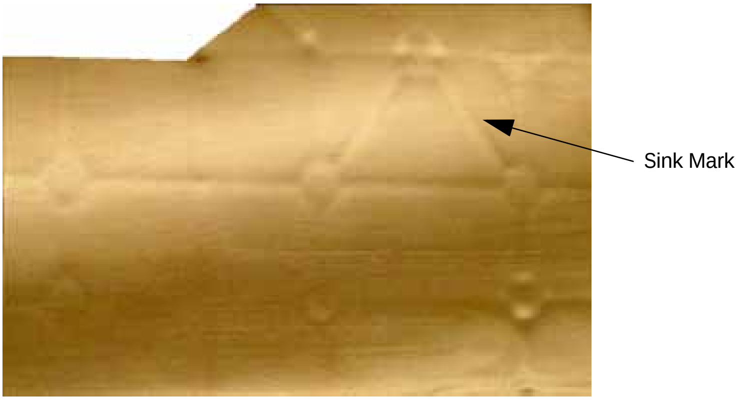


Figure 3.9 Sink marks

3.2.9 Multidirectional Flow

Multidirectional flow is caused by the flow changing direction during filling. This results in orientation in different directions that creates problems with flow marks, stress, warping, and more. As the plastic starts to flow we have a simple flow front as shown in Figure 3.10 (a). When the flow fronts meet the top and bottom edges, however, there is a minor change in flow direction. There is a major change in flow direction when the flow front meets the right edge (Figure 3.10 (b)). The contour lines in Figure 3.10 (b) represent the position of the flow front, while the arrows show the directions of the flow and the orientation. In this example, these changes in direction cause both overpacking and underflow.

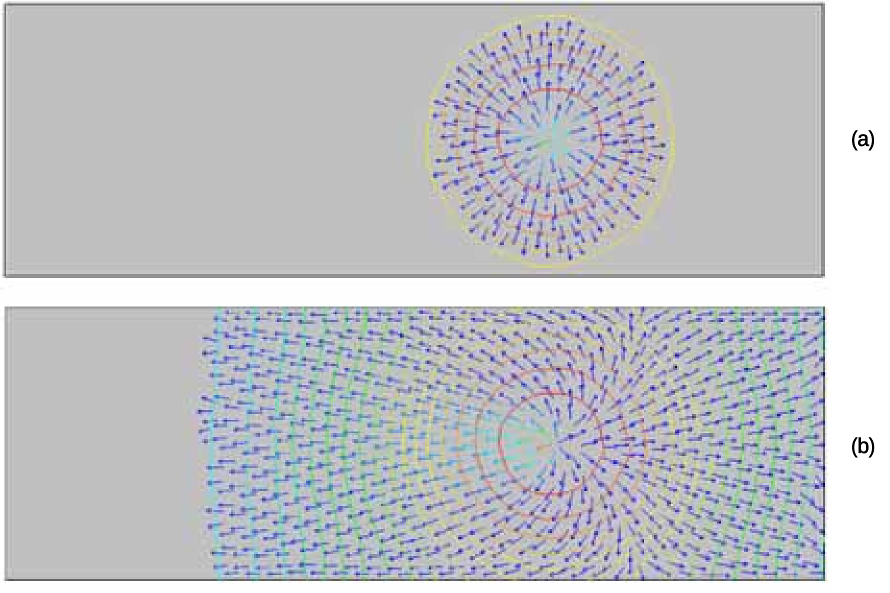


Figure 3.10 Multidirectional flow

3.2.10 Unstable Flow

Plastic flow can at times appear unpredictable because of an instability arising from the combination of heat transfer and fluid flow. Consider the apparently balanced system based on an actual mold shown in Figure 3.11. In practice the filling was unstable. On one shot, cavity A would fill first. On the next shot, B would fill first, then cavity A first again, so each cavity would fill first on alternate shots.

Investigation found that changing the mold temperature by as little as 3°C would cause either cavity to consistently fill first. The instability occurs when a cavity (B, for instance) fills last, then the mold will be a little hotter for that cavity, as there has been less time for the cavity to cool. Therefore, on the next shot cavity B will fill first and A last, making cavity A now hotter, hence giving this consistent instability.

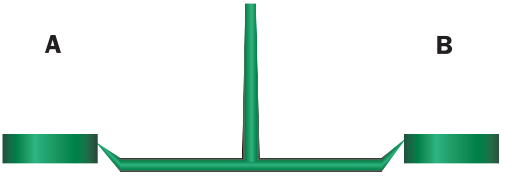


Figure 3.11 Thermally unstable part

3.2.11 Simple Flow Pattern

The essence of mold filling is a simple flow pattern. Complex flow patterns, with changes in direction of flow or variations in flow rate, always reduce part quality. The ideal flow is to have a straight flow front across the mold, giving a uniform orientation pattern. The objective of the Moldflow design procedure is to position gates, dimension the runner system, and possibly modify the dimensions of the part to get a simple flow pattern.

3.3 Flow-front Velocity and Flow-front Area

3.3.1 What are FFV and FFA?

Here we present two simple yet important design and process parameters: flow-front velocity (FFV) and flow-front area (FFA). As the name suggests, flow-front velocity is the polymer flow-front advancement speed through the feed system and cavity. Flow-front area is defined as the cross-sectional area of the advancing flow front. FFA is calculated by multiplying either length of the flow front by the thickness of the part (see Figure 3.12) or the cross-sectional area of the runner, or multiplying by a sum of both, if the flow is active in both places. At any time, the product of local FFV and FFA along all moving fronts is equal to the volumetric flow rate, neglecting material compressibility.

3.3.2 Flow-front Velocity Influences Filling Pattern

3.3.2.1 Constant FFV

The ideal filling pattern has flow front reaching every extremity of the cavity simultaneously with a constant flow-front velocity throughout the process. Otherwise, localized overpacking at prematurely filled regions might occur within the part.

3.3.2.2 Variable FFV

A variable FFV during filling also leads to changes in the molecular or fiber orientation. When the molten plastic contacts the cold mold, it immediately freezes at the part surface region, resulting in varied orienatation. For any mold that has a complex cavity geometry, a constant ram speed (or constant volumetric flow rate) will not have a constant velocity at the advancing flow front. Whenever the cross-sectional area of the cavity varies, part of the cavity may fill faster than other areas. Figure 3.12 shows an example where the FFV increases around the insert, even though the volumetric flow rate is constant. This creates high stress and varied orientation along the two sides of the insert, potentially resulting in differential shrinkage and part warpage.

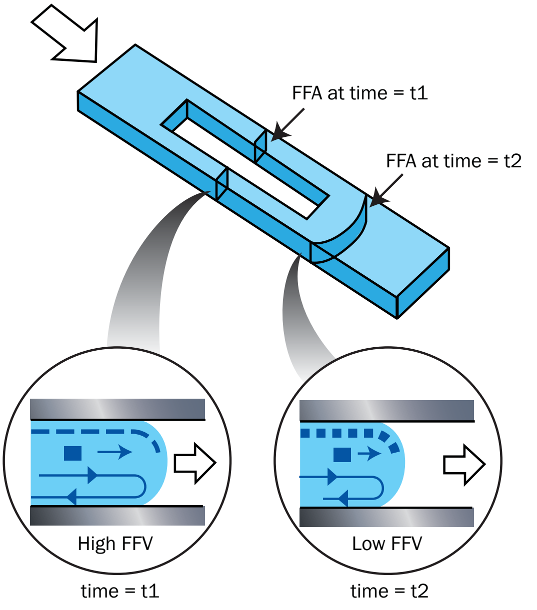


Figure 3.12 Figure 3.45 Flow-front velocity and flow-front area. Note that a constant volumetric flow rate does not necessarily guarantee a constant velocity at the advancing flow front because of the variable cavity geometry and filling pattern. With a variable FFV, the material element (indicated by the squares in the enlargements) will stretch differently, resulting in differential molecular and fiber orientations.

3.3.3 Equation

The relationship of volumetric flow rate, FFA, and an averaged FFV can be expressed as:

$$\text{Flow-front velocity (FFV)} \approx \frac{\text{Volumetric injection flow rate}}{\text{Flow-front area (FFA)}} \tag{3.1}$$

When the volumetric flow rate is calculated based on the molding machines ram velocity, the flow-front velocity will be slightly less than the value calculated in Equation 3.1 due to material compressibility.

3.4 Using Moldflow to Determine the Filling Pattern

3.4.1 Computer Simulation Can Eliminate Molding Trials

Traditionally, the filling pattern of a mold was determined by conducting a series of short shots on the molding floor. This involved running an injection-molding machine with either a prototype or an actual production mold. Now computer simulation or a flow analysis is conducted. A flow analysis of a part is best done early in the product design cycle. The earlier in the design cycle a flow analysis is done, the earlier problems caused by filling the part are found, and the easier—and less costly—they are to fix.

3.4.2 Using a Flow Analysis

3.4.2.1 How It Works

A flow analysis creates many plots including fill time. This shows the filling pattern sometimes called isochrones (contours at equal time intervals). The space between adjacent contours represents the flow-front velocity. Closely spaced contours indicate hesitation, whereas widely spaced contours indicate racetracking.

3.4.2.2 Interpreting the Results

In Equation 3.13, the flow analysis results show the filling pattern on a television front bezel. The fill-time contours indicate that the flow-front velocity is not uniform. The FFV is high near the gates and end of fill are shown by the widely spaced contour lines. Weld-line and air-trap locations are also shown. Although you can use the fill-time result to locate weld/meld lines and air traps, the task will be very difficult for parts of complex geometry. For this reason, Moldflow provides additional capability, specifically for automatic weld/meld line and air-trap prediction.

3.4.2.3 Improving the Filling Pattern

With the aid of a flow analysis, you can improve the filling pattern by changing the:

- Part design
- Tool design
- Gate location
- Injection velocity profile
- Mold and melt temperatures
- Material selection

The advantage of using a flow analysis is you can optimize the design of the part and tool before the tool is done so the part quality will be higher, take less time to get the tool into production, and be less expensive to produce.

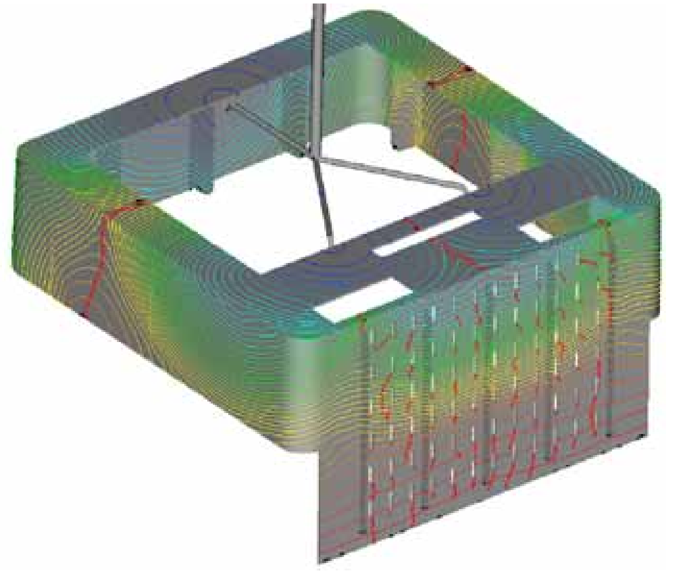


Figure 3.13 TV front bezel with filling pattern, weld lines, and air traps displayed

3.5 Using Moldflow to Achieve Constant FFV

This example illustrates how a flow analysis aids the design process in one aspect. It uses Moldflow to improve a design with the goal of making the flow-front velocity constant. Because the FFV affects the degree of molecular and fiber orientation, it should be kept as constant as possible.

3.5.1 Controlling the FFV Through Ram Speed

One strategy for maintaining a constant FFV during filling is to adjust the ram-speed profile. All modern injection-molding machines have this capability. In this example we compare the flow-front advancement on a center-gated square part with one corner having a longer flow length, creating a part that is not symmetrical. The first analysis uses a constant ram-speed profile, and the second has an optimal ram-speed profile developed by a Moldflow fill analysis. The optimal ram-speed profile takes into account the variation in FFA, which is typical of many cavity part geometries.

Design 1: In Design 1, with a constant ram speed, the polymer melt initially spreads radially from the center gate, resulting in an initial increased FFA (Figure 3.14). Once the flow front reaches the side walls, FFA decreases filling three of the four corners. With only one corner to fill, the FFA rapidly reduces in size.

With a constant ram-speed profile and a variable FFA, the FFV initially decelerates (indicated by the diminishing spacing between adjacent contours), before it shoots up again as the FFA gets smaller. Such a variation in FFV is not desirable, considering the effect of the various molecular or fiber orientation and stress levels it causes. The influence of the FFV on shear stress can be seen in Figure 3.15.

Design 2: Design 2 employs an optimal ram-speed profile recommended by Moldflow (Figure 3.16). The FFV for the same part as in Design 1, above, becomes more uniform during cavity filling. This is shown by the equal spacing in the predicted filling contours. As shown in Figure 3.15, a constant FFV lowers shear stress in the part.

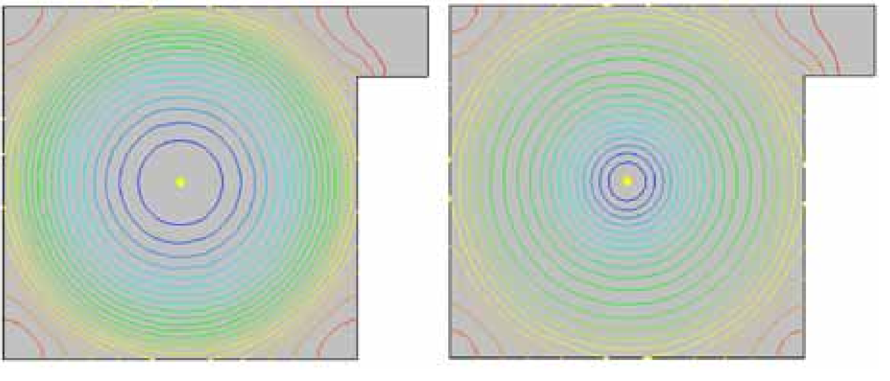


Figure 3.14 Comparison of filling patterns with a constant flow-front velocity and constant ram velocity (constant volumetric flow rate)

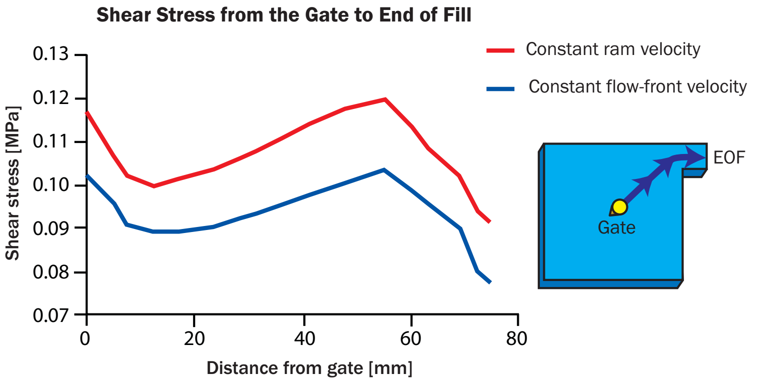


Figure 3.15 Comparison of shear stress in the part from the gate to the end of fill (EOF) with a constant ram velocity and constant flow-front velocity

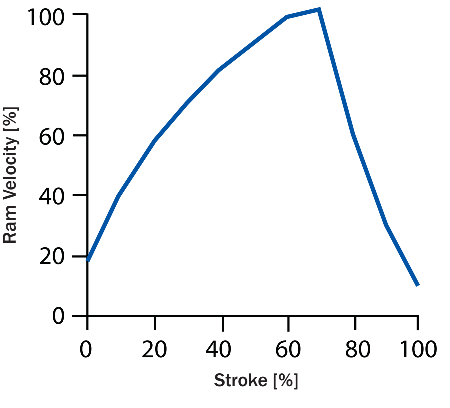


Figure 3.16 Ram speed profile recommended by Moldflow

4 Moldflow Design Principles

- Product design and Moldflow
- Sequence of analysis
- Moldflow flow concepts

4.1 Product Design and Moldflow

Like all design procedures, these stages are based on the ideal. In practice compromises may need to be adopted. The analysis for some parts may show that the compromise is not acceptable (i.e., it will not give workable parts of the required quality), in which case the whole part design can be reviewed with the product designer before locking into a disastrous course. A small change in product design can often give dramatic overall improvements. This integration of mold and product design is a key element in Moldflow's design principles.

4.2 Sequence of Analysis

The procedure for mold design always starts with the cavity by testing options for gate positions, optimizing molding conditions within the cavity, then using that gate layout and making corrections, until the cavity conditions are acceptable. Then, using these conditions the procedure addresses upstream tasks, such as defining runner dimensions. In this procedure, optimizing molding conditions is a key part of the design process.

Once the gate position has been fixed and the molding conditions established, the flow rate, melt temperature, and required pressure in the runner system are determined. In other words, the cavity analysis determines a specification for the runner design.

Once the filling of the part has been optimized, the cooling system for the part can be analyzed. Generally the goal is to design the cooling system of the mold to uniformly extract heat from the part. This will minimize the cycle time while producing high quality parts.

Even though filling and packing are closely related, packing is best optimized after the cooling analysis. The packing and compensation phases are dominated by heat transfer, while filling is dominated by fluid flow. The cooling analysis provides an accurate picture of how the part's heat is extracted, so it is best to optimize the packing of the part after the cooling.

The final step is to determine the warpage of the part. When the part is properly analyzed in the previous steps, the warpage analysis is a confirmation that the part and process optimization is well done.

4.2.1 Part Filling Optimization

A key component of an injection-molded product analysis is the part itself. The part analysis should start early in the design stage. Below is a description of the three main steps for part filling optimization.

4.2.1.1 Determine the Number of Gates

This is primarily driven by pressure requirements. The pressure required to fill the part should be well under the capacity of the machine. A conservative guideline is that the fill pressure for the part should be half the machine pressure. For a typical machine, this is about 70 MPa (10,000 psi). The limit is about half the machine limit because at this stage of the design process, the pressure drop through the runners is not being calculated. The total pressure drop for the entire tool (parts and runners) should be about 75% of the machine capacity at maximum.

Use as few gates as possible. One gate is normally best. Add gates as necessary to reduce the pressure to fill or to achieve a desired fill pattern.

4.2.1.2 Position the Gates for Balanced Filling

The gate position should produce a balanced flow front within the part, with no underflow or overpacking effects. If the filling pattern cannot be balanced by changing the gate position, flow leaders or deflectors can be used to balance the flow.

4.2.1.3 Ensure the Flow Pattern Is Unidirectional

The filling pattern should be straight and uniform. In addition, there should be no problems with hesitation, underflow, weld lines, air traps, etc.

4.2.2 Molding Conditions

The molding conditions used must be determined in conjunction with the steps above. When considering the number of gates, the pressure required to fill the part must be considered. The molding conditions used, mold temperature, melt temperature, and injection time can make a huge difference in pressure. A molding window analysis can quickly evaluate a gate location to determine if it is possible to use the number and position of gates specified. The molding window should also be as large as possible. Having a large molding window allows a wide variation in molding conditions while still producing a good part.

4.2.3 Runner Design

The runners should be designed to aid in achieving the required flow pattern. Runners may need to be sized to achieve the desired filling pattern on larger multigated parts. They should be balanced and have minimal volume. The runners must also be designed to achieve the molding conditions that were used to optimize the part. Generally to account for the shear heat generated in the runner system the melt temperature entering the sprue must be lower than that entering the gates. Also the injection time must be increased to allow time to fill the runners. It is best to have the same flow rate filling the part with runners as there was in the analyses without the runners.

4.2.4 Cooling Optimization

The design of the cooling system can be optimized once the part and runner system have been analyzed. The part and runner system provide the heat input into the tool. The cooling analysis aids the design of a proper cooling system. By minimizing the cycle, heat extraction from the part is balanced so heat is transferred equally out of both sides of the plastic cross section. This will minimize warpage.

4.2.5 Packing Optimization

During the packing and compensation phases, the volumetric shrinkage within the part is determined. The level of volumetric shrinkage is a key factor in the shrinkage and warpage of the part. Packing analysis is best done after the cooling has been optimized because heat transfer dominates during packing and compensation.

4.2.6 Warpage Optimization

The final step in the part analysis process is to look at the warpage of the part. Warpage is influenced by material selection, part design, tool design (runner and cooling system design), and molding conditions, all of which are considered when following the sequence described previously. When filling, cooling, and packing are properly optimized, the warpage analysis becomes a validation that the product, tool, and process have been optimized. When the warpage analysis indicates the part is out of tolerance, it also indicates the major cause of the warpage so further optimization can address the cause.

4.3 Moldflow Flow Concepts

The Moldflow flow concepts are a set of rules that influence both the design of a part and tools to optimize the part's filling. When these principles are followed, higher quality parts and faster cycle times are the result. Not following the principles leads to problematic designs.

Table 4.2: Moldflow flow concents

Unidirectional and controlled flow pattern	Balancing with flow leaders and flow deflectors
Flow balancing	Avoid underflow
Constant pressure gradient	Avoid hesitation effects
Maximum shear stress	Controlled frictional heat
Uniform cooling	Thermal shutoff of runners
Positioning weld and meld lines	Acceptable runner/cavity ratio

4.3.1 Unidirectional and Controlled Flow Pattern

To produce unidirectional orientation, the filling pattern in the part should be unidirectional (i.e., it should not change directions during the filling phase). In Figure 4.1 a rectangular part is shown gated in one of two places. The top example is center-gated top to bottom, but is closer to the right edge than the left. The bottom part is gated on the right edge. Figure 4.2 shows the flow direction arrows laid over the fill contour lines. The contour lines show the position of the flow front as the part is filling. The flow direction arrows show the flow direction when the part is nearly filled. The contour lines and arrows should be perpendicular to each other. With the center-gated part, the flow has significantly changed directions causing a nonuniform direction of the orientation within the part. The end gated part has no change in direction.

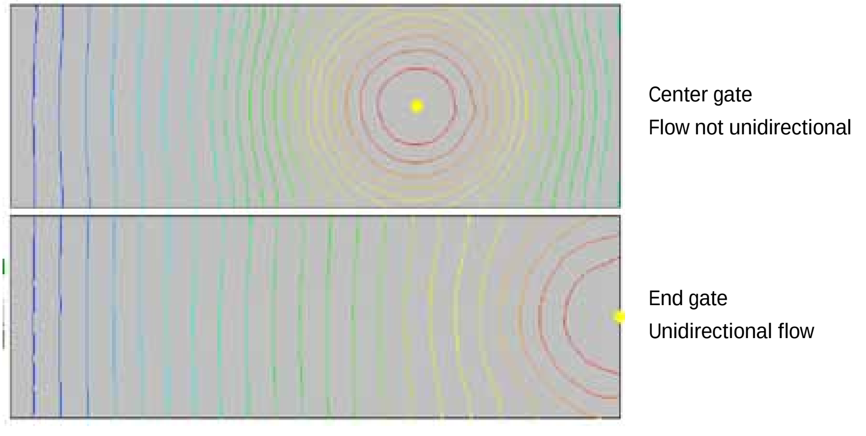
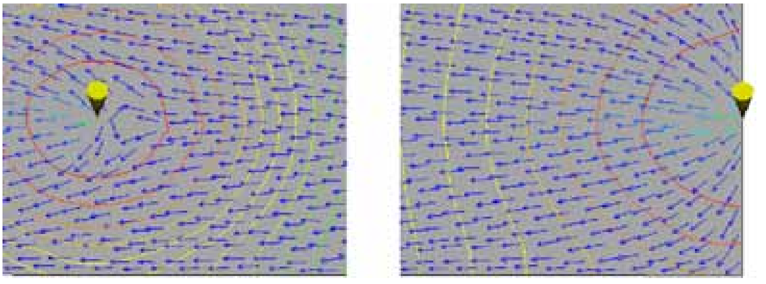


Figure 4.1 Gate location influences flow direction



Center gate
Flow direction changes
after flow front hits edges of part

End gate
Flow direction unchanged
during entire filling

Figure 4.2 Flow pattern changes with gate location

4.3.2 Flow Balancing

All the flow paths within a mold should fill at the same time and with equal pressure. For multicavity molds, this means each cavity should fill at the same time. Within parts the same holds true: the extremities of the part should also fill at the same time.

4.3.2.1 Runner System Balance

There are two types of balanced runner systems: naturally balanced, sometimes called geometrically balanced runners, and artificially balanced runners. In a naturally balanced runner system, the flow length from the sprue to each of the parts is the same for all cavities, as shown in Figure 4.3. Generally this type of runner system has a larger processing window than artificially balanced runners.

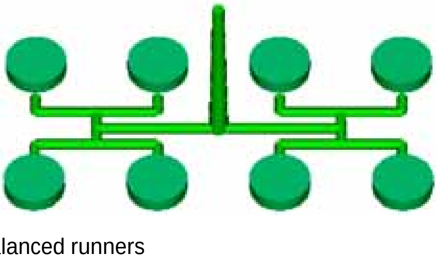


Figure 4.3 Naturally balanced runners

The artificially balanced runner system achieves its balance by changing the size of the runners. This can be a very useful technique for balancing runners, as there is generally less runner volume required than for a naturally balanced runner. However, because of the runner diameter, the molding window is generally smaller than a naturally balanced runner. Injection time is generally the main limiting factor. Figure 4.4 shows an example of an artificially balanced runner system.

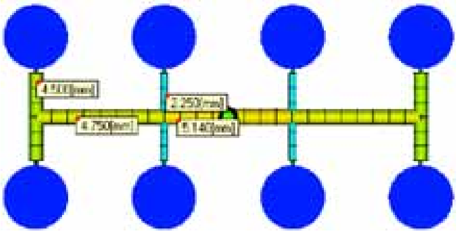


Figure 4.4 Artificially balanced runners

4.3.3 Constant Pressure Gradient

The pressure gradient while the part is filling should be uniform through the part. Figure 4.5 shows a part that does not have a constant pressure gradient during filling. The XY graph is the pressure at the injection location. Just at the beginning of fill, there is a spike in pressure. However, the big problem is at the end of fill. The part is filling mostly by radial flow. As the flow front meets the center of the sidewalls, the flow front starts contracting. This corresponds to a slight increase in the pressure gradient. The big spike occurs when the three corners fill and the remaining upper right corner is the only unfilled area remaining. All the material exiting the gate enters the upper right corner, causing the pressure spike. The volumetric flow rate entering the part is constant. The pressure gradient is an indication of a balance problem, or it suggests an injection velocity profile should be used.

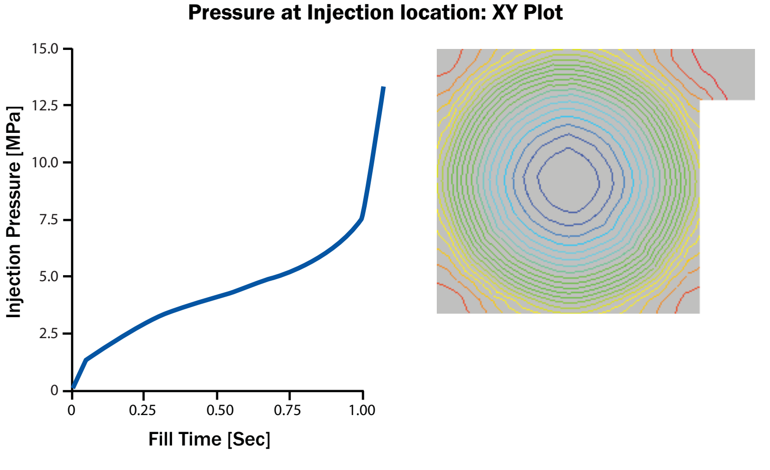


Figure 4.5 Pressure gradient

4.3.4 Maximum Shear Stress

The maximum shear stress in the part should be below the material limit specified. The shear stress limit for the material can be found in Moldflow's material database. The shear stress limit is approximately one percent of the tensile strength of the material and is also application-specific. For parts used in harsh environments such as elevated temperatures, under a high load during use, or exposed to chemical attack, the limit specified in the database may be too high. Alternatively, if the part is not used in a harsh environment the limit is conservative (low), and the stress can be significantly exceeded without any problems. When the shear stress does get above the limit, however, it should be kept as low as possible.

Figure 4.6 shows the maximum shear stress in the part scaled from the material limit to the maximum shear stress value calculated in the analysis. Areas that are colored in the plot are therefore above the limit. In this case, the maximum shear stress is 0.45 MPa, which is not too high. Most of the time, parts will have areas of high shear stress that will be two to five times the stress limit. In this case, it is only 1.5 times the limit. However, much of the part is slightly above the limit. The maximum shear stress in the cross section is at the frozen/molten layer interface, or *at wall*.

Three main factors influence shear stress:

- Wall thickness—increase the wall thickness to reduce stress
- Flow rate—lower the flow rate (locally or globally) to reduce stress
- Melt temperature—increase the melt temperature to lower the shear stress

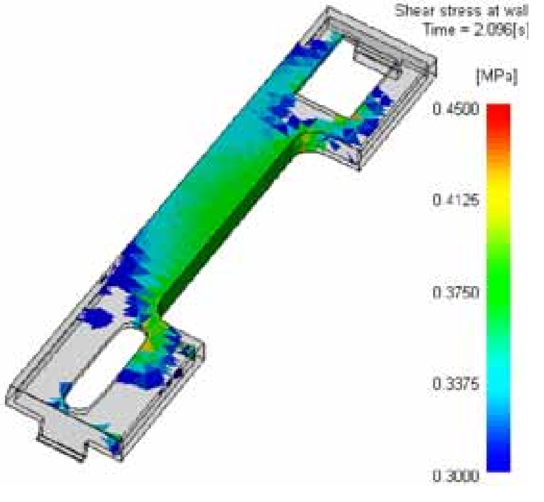


Figure 4.6 Maximum shear stress

4.3.5 Uniform Cooling

When cooling a part, the mold surface temperature should be uniform on both sides of the part. When the temperatures are not uniform, the molecules on the hot side have a longer time to cool and thus shrink more. This makes them shorter, so the parts will bow toward the hot side of the part as shown in Figure 4.7.

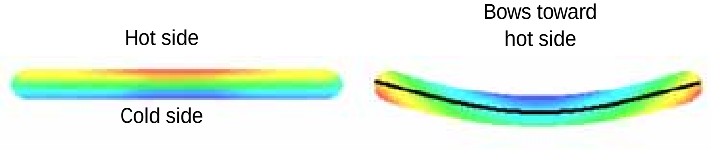


Figure 4.7 Uniform cooling

Figure 4.8 shows the typical box-type structure of many injection-molded parts. In the box structure, there is an inside corner (the core) that is normally difficult to cool and where heat tends to concentrate. The cavity side is easy to cool, and there is a larger volume of mold to absorb the heat from the plastic. As a result, the inside of the corner runs hot, allowing more time for the molecules to cool down and shrink, therefore collapsing the corner a bit. This will pull the sides of the box toward the core.

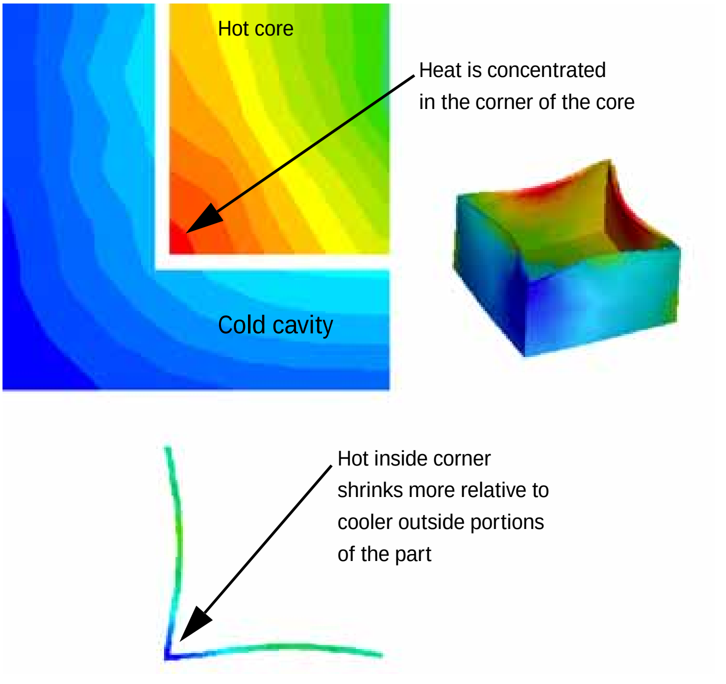


Figure 4.8 Cooling of box structures

4.3.6 Positioning Weld and Meld Lines

A weld line is formed when two flow fronts meet head-on. A meld line is formed when the flow fronts meet while flowing in the same direction. Formation of weld lines and meld lines is shown in Figure 4.9. Weld lines are generally weaker and more visible than meld lines, but they should both be avoided.

Every time a gate is added to the part, an additional weld or meld line is formed, so eliminating extra gates is advisable. When the number of weld or meld lines cannot be reduced, they should be placed in the least sensitive or least critical areas in regard to their strength and appearance. Depending on the application, a weld or meld line could be a problem in terms of either strength or appearance. The strength of weld or meld lines generally is improved when formed at higher temperatures and when the pressures to pack them out is higher. Venting is also important. Weld lines should be vented to maximize their strength and minimize their appearance.

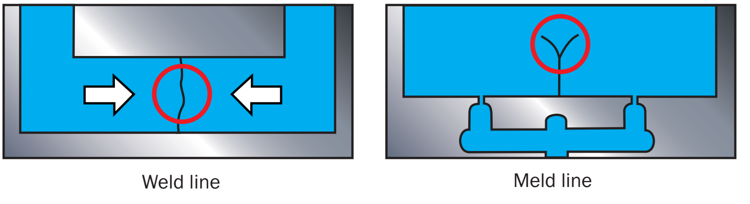


Figure 4.9 Weld line and meld line

4.3.7 Avoid Hesitation Effects

Hesitation is an unintended slowing down of the flow front. When a flow front slows down too much, it gets too cold and in severe cases can freeze off. This is what has happened in the top example in Figure 4.10. Hesitation will occur when there is a large variation in wall thickness in the part. In this case, the rib is much thinner than the nominal wall. Having a fast injection time can minimize hesitation because by increasing shear heating, there is less time for the material to hesitate. Another way to reduce hesitation is to gate as far as possible from thinner areas, as was done in the bottom example in Figure 4.10.

4.3.8 Avoid Underflow

Underflow occurs when a flow front changes direction during filling. In the example in Figure 4.11, underflow occurs because the flow front is not balanced due to the gate location. The contour lines and the velocity arrows should be perpendicular to each other. On the right side of the magnified area, the contour lines and velocity arrows are parallel, indicating a significant shift in the flow direction.

The problem with underflow is its effect on orientation. The initial filling direction for an area on the part is represented by the fill-time contours. The flow direction is perpendicular to the contour line. The molecules are initially oriented in the direction of that flow. If the flow direction changes later on during the filling phase, the molecules closer to the center of the flow channel are oriented in the new flow direction. Molecules generally want to shrink more in the direction of orientation, so there is significant internal stress in locations where underflow occurs.

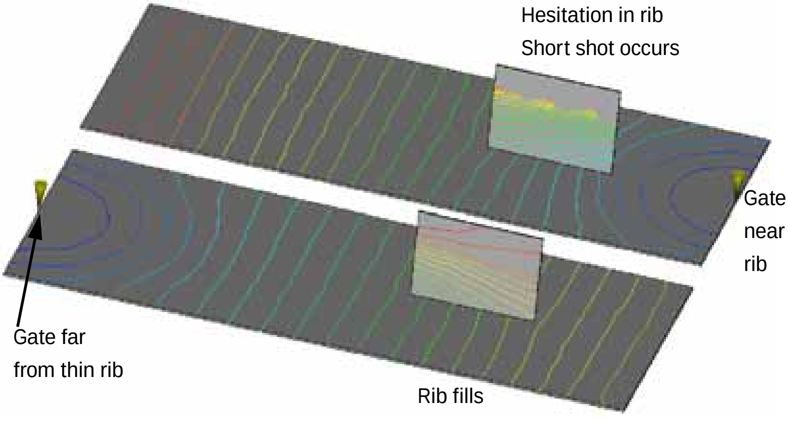


Figure 4.10 Hesitation

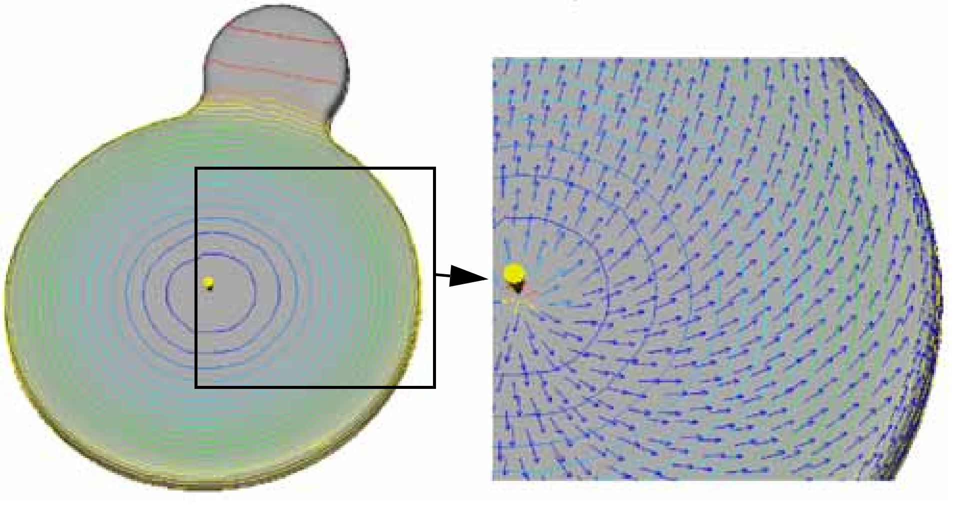


Figure 4.11 Underflow

4.3.9 Balancing with Flow Leaders and Flow Deflectors

Flow leaders are local increases in the nominal wall thickness, whereas flow deflectors are local decreases in thickness.

Many times a part cannot be balanced by gate placement alone. It may be useful to slightly change the wall thickness to enhance or retard the flow in a certain direction. This will allow the filling of the part to be balanced, even though the flow lengths from the gate to the extremities of the part are not equal. Figure 4.12 shows an example of using flow leaders to balance the filling of a rectangular-shaped part to achieve a balanced fill. The original part has a nominal wall of 1.5 mm, but has a thick rim. Because of the part's shape and the gating location, the flow front races around the part once the material hits the thick rim. This causes air traps and weld lines on the ends of the part in addition to its not being balanced. By adding flow leaders, the filling was balanced, removing the weld lines and air traps. Generally it is better from a material saving point of view to decrease the wall thickness. This may not always be possible, however, due to structural requirements of the part.

The change in wall thickness preferably should be gradual. The change in wall thickness should be no more than about 25% of the nominal wall. Large changes in wall thickness may lead to cooling and orientation problems, and may increase the warpage rather than decreasing it.

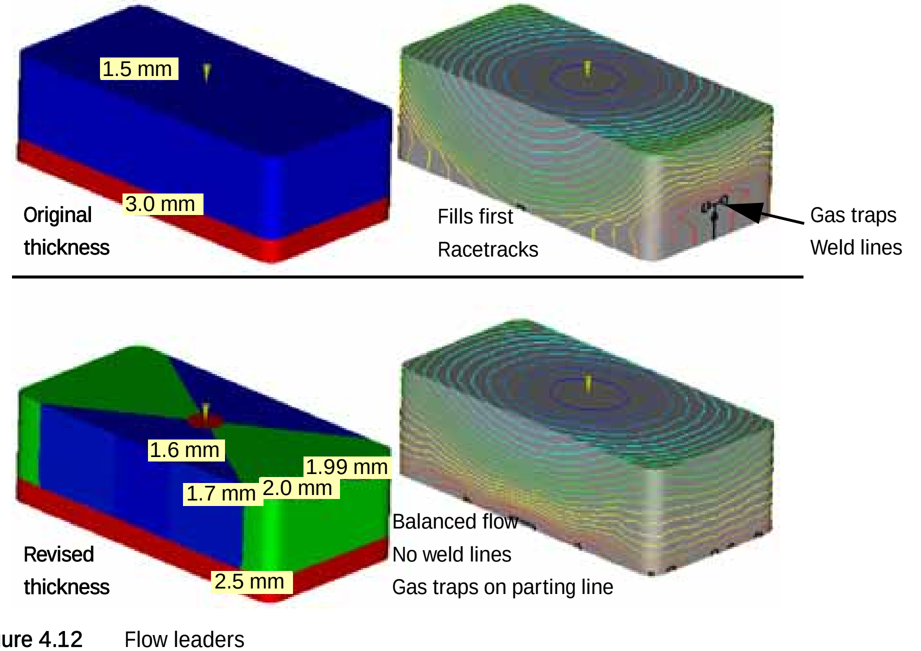


Figure 4.12 Flow leaders

4.3.10 Controlled Frictional Heat

Runners should be sized so they produce frictional (shear) heat. When runners are properly sized and balanced, the temperature entering the part should be within 2 to 3ºC of the optimum melt temperature determined for the part. The temperature entering the sprue is typically 10 to 30ºC below the temperature entering the part. Figure 4.13 shows a balanced runner system of a family tool. Because of the balance, the temperature entering the parts is nearly identical, even though the runners are different lengths.

The advantages of shear heat in the runner system are as follows:

- Allows for higher melt temperatures to fill the part
  - Reduces pressure to fill the part
  - Reduces shear stresses in the part
  - Easier to pack the part
- Longer residence time in the machine barrel

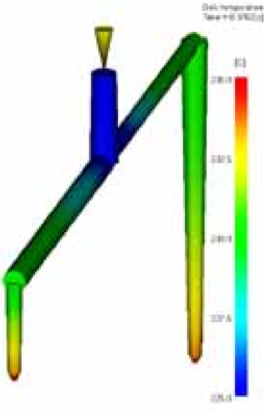


Figure 4.13 Controlled frictional heat

4.3.11 Thermal Shutoff of Runners

The runners should be sized so they allow the parts to fill and pack out without controlling the cycle time. In Figure 4.14 the freeze time of the part is about 3.4 seconds. The runners have freeze times that are at least that of the part. Notice, however, that the cooling time of the sprue is about 10 times that of the part. This would suggest the sprue is too large and should be made smaller if possible. The largest cooling time in a runner should preferably be at most two to three times that of the part, but this is often difficult to do. In the case of the runner in Figure 4.14, if the runners were made smaller while maintaining a balanced runner system, the smallest runner, which currently has a cooling time of 4.7 seconds, would quickly become much smaller than the part. As a general rule, if there are no critical dimensions or sink mark quality criteria, the cooling time of the runners can be as low as about 80% of the cooling time of the part. When dimensions are more critical, the cooling time for the runners should be greater than the part.

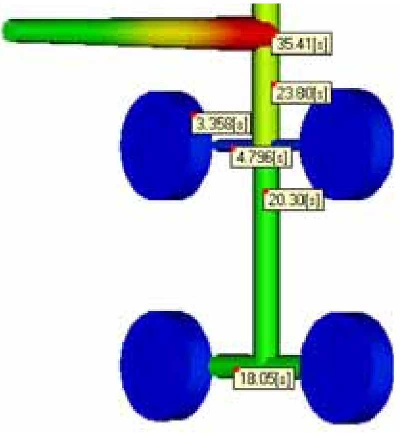


Figure 4.14 Thermal shutoff

4.3.12 Acceptable Runner/Cavity Ratio

The ratio of the volume of the runner system to the total volume of the cavities should be as low as possible. This is to reduce the material being wasted in the runners and to reduce the amount of regrind. In Figure 4.15, the runners cannot be made much smaller and still maintain a balanced fill and acceptable packing. In this example, the ratio of runner to cavity volume is 85%, which is very high. Ideally, the volume of the runners should be 20% of the part volume or less. In this example, the volume of the sprue is quite high. A hot sprue can be used to reduce the volume of the cold runners.

Hot runner systems should also have volumes smaller than the part. This reduces both the residence time the amount of compressibility in the runner system.

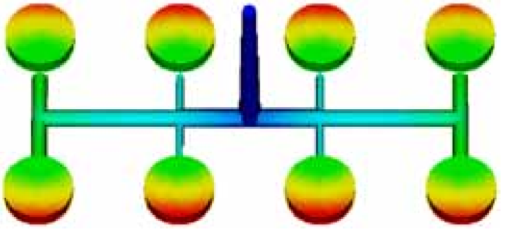


Figure 4.15 Cavity/runner ratio

5 Meshes Used In Moldflow Analyses

- Mesh types used by Moldflow
- Mesh requirements
- Geometry creation
- Importing geometry
- Using different mesh types

5.1 Mesh Types Used by Moldflow

5.1.1 Finite Elements Used in Moldflow

In order to run a Moldflow analysis, the part model must have an appropriate finite element mesh created on it. Often, the finite element mesh is referred to simply as a *mesh*. Elements divide the geometry (domain) of the part or other tool component into a number of small domains. These small domains or elements are defined by nodes (coordinates in space) and are used for the calculations inside Moldflow. There are three main categories of elements:

- **Beam**: two-noded element used to describe the feed system, cooling channels, etc.
- **Triangle**: three-noded element used to describe the part, mold inserts, etc.
- **Tetrahedron**: four-noded element used to describe the parts, cores, feed systems, etc.

Examples of these three element types are shown in Figure 5.16.

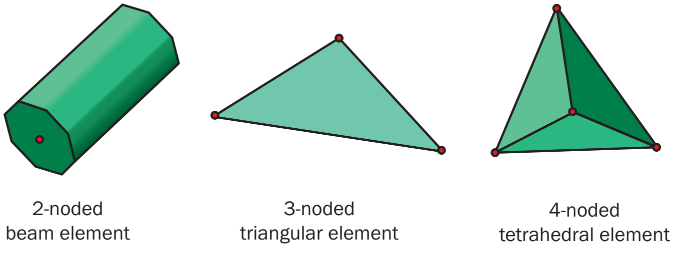


Figure 5.16 Element Types

5.1.2 Mesh Types

Moldflow uses three mesh types for analysis. The mesh types use a combination of the element types described above. The mesh types are:

- Midplane
  - The mesh is defined on the *midplane* or centerline of the plastic cross section as shown in Figure 5.17 (a).
  - Triangular elements are primarily used to define the part.
  - Beam elements can be used to define the feed system, cooling channels etc.
- Fusion
  - Triangular elements are defined on the surface of the plastic cross section as shown in Figure 5.17 (b).
  - Analysis method called Dual Domain™.
  - Beam elements can be used to define the feed system, cooling channels etc.
- 3D
  - Tetrahedral elements are used to represent the part. Several rows of elements are used to define the cross section as shown in Figure 5.17 (c).
  - Beam elements or tetrahedral elements can be used to represent the feed system.

Care should be used when using the term "*mesh*." Depending on the context, it could be referring to a collection of a certain type of finite element, e.g. a "triangular mesh," or it could mean a type of analysis, e.g. "A midplane mesh was used."

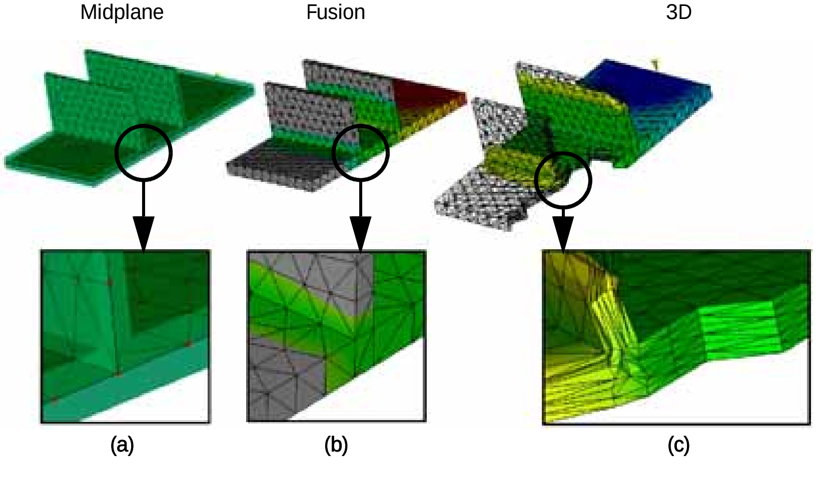


Figure 5.17 Mesh types

5.1.3 Solver Assumptions

5.1.3.1 Midplane and Fusion

The same flow solver is used for Midplane and Fusion mesh types. Every type of solver has certain assumptions. For midplane and Fusion, the solvers are based on the generalized Hele-Shaw flow model. This model has the following assumptions:

- Laminar flow of a generalized Newtonian fluid
- Inertia and gravity effects can be ignored
- In-plane heat conduction is negligible compared to conduction in the thickness direction
- Thermal convection in the thickness direction is neglected
- Heat loss from edges can be ignored for the triangular element type

Element-specific Assumptions

- Beams: Sometimes referred to as *1D elements*, beams have an assigned cross-sectional size and shape. Beams represent axisymmetric circular-tube flow of a generalized Newtonian fluid. A noncircular shape typically is represented by an equivalent circular tube with the same hydraulic diameter, but with the volumetric flow rate scaled down in order to give the same average velocity as the original shape. Juncture losses from abrupt contractions in the flow path are incorporated through an empirical model derived based on Bagley corrections in viscosity characterization. Beam elements cannot account for shear-induced imbalances, as sometimes seen in feed systems.
- Triangles in midplane meshes: Triangular elements used in a midplane mesh are often referred to as *2.5D elements* or *shell elements*. This mesh simulates a 3D part with a two-dimensional plane surface at the center of the thickness. A thickness property is assigned to this plane, hence the terminology 2.5D. Because of the assumptions listed above, the cross section that can be modeled with this element type is limited. As a minimum, the width to thickness ratio of any local area should be at least 4:1, otherwise significant errors may be introduced. At a 4:1 width to thickness ratio, 20% of the perimeter is in the thickness direction and is not accounted for in the heat transfer equations. The greater the violation of this rule, the greater is the amount of possible error. This is a particular problem for square-shaped geometry, such as connecting ribs, housing vents, or grills.
- Triangles in Fusion meshes: A Fusion mesh, sometimes called a *modified 2.5D mesh*, simulates a 3D part with a boundary or skin mesh on the outside surfaces of the part. The main difference between midplane and Fusion meshes is how the thickness is determined. In Fusion meshes, elements across the thickness are aligned and matched. The distance between the elements on the opposite sides of the wall defines the part thickness. The mesh density is an important factor in determining the accuracy of the thickness representation, in particular on tapered features such as ribs. The percentage of matched elements in the Fusion mesh is a key factor in determining the quality of the mesh. (It should be at least 85%.)

5.1.3.2 3D Meshes

A 3D mesh makes fewer assumptions than midplane and Fusion meshes. 3D meshes:

- Use full 3D Navier-Stokes solvers
- Solve for pressure, temperature and the three directional velocity components at each node
- Consider heat conduction in all directions
- Provide options to use inertia and/or gravity effects

3D meshes create a true 3D representation of the part. A 3D mesh works well with "thick and chunky" parts that violate the thickness rules for midplane and Fusion meshes, such as electrical connectors and thick structural components.

5.2 Mesh Requirements

In addition to having a mesh with no errors in it, the mesh should also represent the part correctly. The mesh density is an important consideration in addition to properly representing the geometry of the part.

5.2.1 Mesh Density Considerations

Generally, it is easy to achieve a mesh density that can provide good pressure predictions. It does not take a fine mesh to accurately predict pressures. Filling effects, however, can only be accurately predicted if the mesh is detailed enough to capture relevant details of the model. Three important considerations include:

- Hesitation
- Air traps
- Weld lines

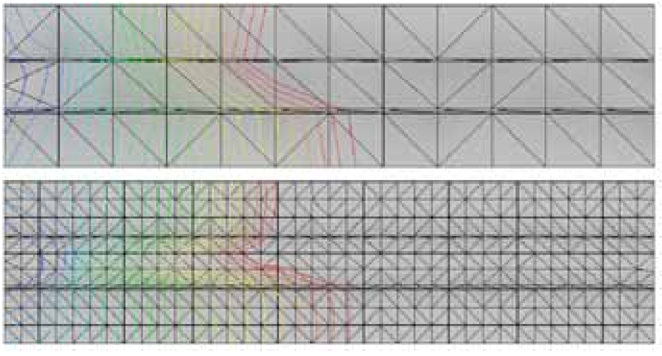
These issues represent common mesh density-related problems. If the mesh is not fine enough, the analysis will not pick up these problems.

5.2.1.1 Hesitation Prediction

Hesitation is a slowing down of one area of the flow front compared to another. To some degree, a small amount of hesitation can be designed into the mold, as is done when flow deflector or artificially balanced runners are used. However, to pick up these or any other type of hesitation effects, a fine mesh is required. Figure 5.18 (a) shows the effect on the predicted filling pattern when the mesh is not fine enough. The center section of the part is 1 mm thick, the top is 2 mm thick, and the bottom is 3 mm thick. Clearly, with the coarse mesh there is no lagging in the thin middle section.

In Figure 5.18 (b), there are three rows of elements across each change in thickness. A much better hesitation pattern is evident in the predicted flow front.

To ensure that hesitation effects are correctly predicted, there should be at least three rows of elements across any major change in thickness.



(a) Filling pattern with coarse mesh

(b) Filling pattern with fine mesh

Figure 5.18 Mesh density influences hesitation prediction

5.2.1.2 Air-trap Prediction

Air traps on a part are often caused by hesitation from changes in wall thickness. The prediction of air traps will only be as good as the mesh density allows. With a coarse mesh in a thin area, air traps will not be predicted or displayed. A fine mesh, however, does allow an air trap to be predicted. In Figure 5.19 the nominal wall is 2.5 mm and the thin wall is 1.25 mm. Notice with the coarse mesh in Figure 5.19 (a), no hesitation is predicted in the thin section. This is shown by the relatively straight contour lines through the thin area. In Figure 5.19 (b), hesitation is predicted with the fine mesh, causing an air trap. An air trap is shown by a colored line around a node on the mesh.

To ensure that air traps are correctly predicted, there should be at least three rows of elements in thin areas of the part.

5.2.1.3 Weld-line Prediction

Weld lines are formed at nodes. When a weld line is predicted at two or more connected nodes, a line is drawn between the nodes. Weld-line prediction is very sensitive to mesh density issues. Therefore, when weld-line information is required, a fine mesh is essential because a coarse mesh does not always indicate the presence of weld lines. In Figure 5.20 (a), the fill-time contours show a V-shaped flow front on the right-hand side of each of the holes, however, no weld line is predicted for the right hole at the end of fill. A weld line will always form when there is a hole in the flow front. With the fine mesh in Figure 5.20 (b), the weld lines are displayed as expected. Notice how the weld line on the right hole in Figure 5.20 (b) is angled up. The locations of the nodes on the mesh influenced the exact position of the weld line.

When reliable prediction of weld lines is critical, ensure those areas of the part where they are most likely to occur are finely meshed.

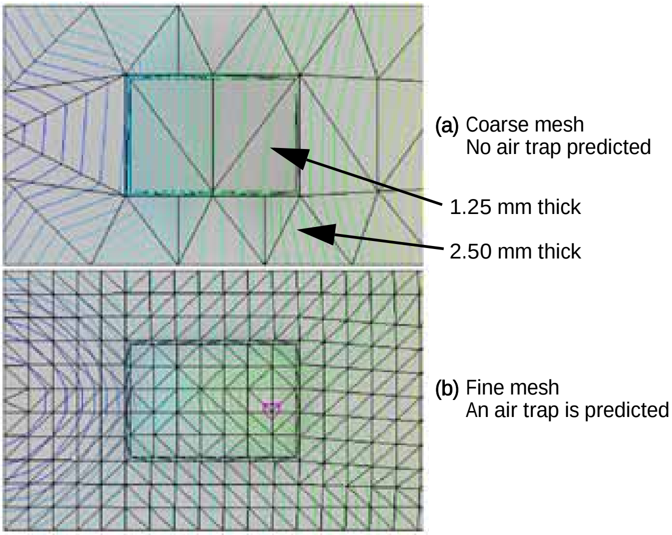


Figure 5.19 Air trap prediction with thick area around a thin area

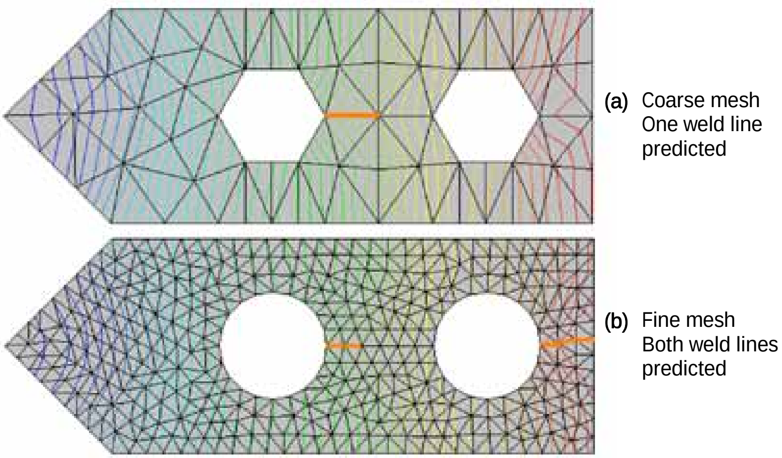


Figure 5.20 Weld line prediction with different mesh densities

5.2.2 Part Details

To properly represent a plastic part for flow analysis, there are three characteristics of the part that need to be modeled accurately:

- Thickness
- Flow length
- Volume

When these part characteristics are correctly modeled, the flow analysis will be accurate.

5.2.2.1 Thickness

The wall thickness of the plastic part is the largest contributor to a pressure drop in the part. Wall thickness is the most critical characteristic of the part design to model for flow analysis.

For midplane models, each element must have a defined thickness. Care must be taken so the thickness is properly set for the elements.

For Fusion models, the thicknesses are calculated automatically by default. The distance between matched elements determines the thickness. To account for extra heat transfer along the part's edge—called edge effects—elements on the edge are set to 75% of the thickness of face elements touching the edge. The analyst must ensure the part's thickness is properly represented with the Fusion model. If it is not, the model must be corrected by manually setting the thickness of the incorrect elements.

For 3D models, the thickness is defined by the imported geometry or mesh. It is not possible to automatically check or correct the thickness in Moldflow, but the measuring and cutting plane tools can be used to estimate the thickness in a local area. However, 3D is the best solver at representing wide variations in thickness that often occur in complex geometries.

The wall thickness of the plastic part is the largest contributor to the pressure drop in the part.

5.2.2.2 Flow Length

The part's flow length is the second most important characteristic to model for flow analysis. The combination of wall thickness and flow length will determine the pressure required to fill a part. The flow length of the part is determined by the analysis itself, not the user.

5.2.2.3 Volume

The volume of the part is calculated from the part shape, size, and wall thickness. Volume is a good way of determining if your model thickness is accurate. Compare the CAD volume to the Moldflow model volume. Normally, the target is for the calculated volume to be within five percent of the true volume. The calculated volume for Fusion models will generally be more accurate than midplane due to the surface mesh used with Fusion. The volume is important as it helps define the flow rate needed in the part, and will significantly influence the pressure calculations in the runner system. The volume of the part has little influence on the pressure drop within the part itself. When the runner system is added, the part volume will influence the flow rate in the runner system and, therefore, the pressure drop.

5.2.2.4 Comparing Thickness and Flow Length vs. Pressure

The graph in Figure 5.21 summarizes the results from a series of analyses where the thickness and flow length were changed to test the effect on pressure. The parameters where changed in increments of 20%. The thickness was reduced from 4.5 mm to 1.8 mm. The flow length was increased from 100 mm to 180 mm. The parameters were changed so the pressure would increase from the base model. In each case, the volume of the part stayed the same, 11.25 cm^3. The cross section was rectangular, so to adjust the volume of the part, the width of the cross section was changed. The material was a nylon, and the processing conditions did not change for any of the analyses.

It is clear from the graph that thickness has by far the greatest influence on the percent change in pressure. The differences in percent change between flow length and thickness may change a little with different processing conditions and materials, but thickness will always have the most effect.

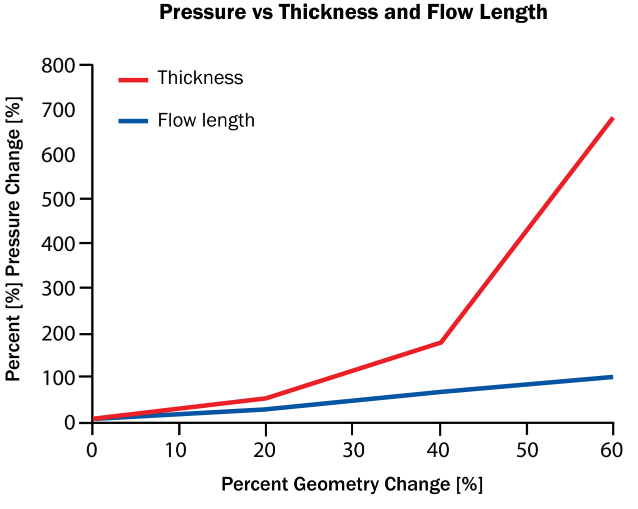


Figure 5.21 Effect of thickness, flow length and volume on pressure

5.3 Geometry Creation

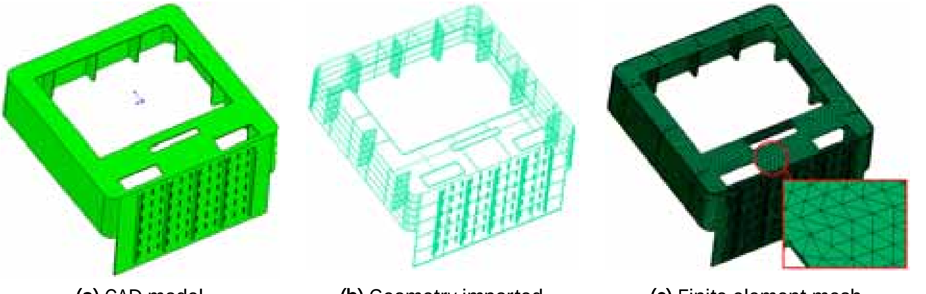
Models that represent the injection-molded part for analysis in Moldflow are generally created in CAD systems as 3D solid models. These models are imported into Moldflow in their native format or by some neutral file format.

In some cases, the finite element mesh needed for the analysis is created in the CAD system or other mesh creation program.

5.4 Importing Geometry

Moldflow supports many formats for importing. The supported formats fall into two main categories: geometry and mesh.

In most cases, the part's geometry is imported into Moldflow, and the finite element mesh required is created within Moldflow. This sequence is shown in Figure 5.22. In some cases, the finite element mesh is created elsewhere and is directly imported into Moldflow.



(a) CAD model **(b)** Geometry imported **(c)** Finite-element mesh

Figure 5.22 CAD model to mesh

5.5 Using Different Mesh Types

When a part needs to be analyzed and there is a choice in the mesh type that can be used among midplane, Fusion, and 3D, which one should be used? Many times part geometries can be used with all three mesh types. Some parts should be modeled with 3D. Following are some examples of determining the appropriate mesh type for different geometries.

5.5.1 Door Panel

The part in Figure 5.23 is a large door panel for a truck. The part has a fairly uniform wall thickness of 3.0 mm (0.118 in), except in the grill area. Section A-A goes through a part of the grill. Both midplane and Fusion can very easily model the geometry over most of the part. However, the grill has a cross section that does not follow the 4:1 width to thickness ratio recommended for midplane and Fusion meshes. In section A-A, the distance between points

1 and 2 is 1.52 mm (0.060 in) and the distance between points 1 and 3 is 8.05 mm (0.318 in). This portion of the rib cross section does have a thickness to width ratio, but the top portion does not. The distance between points 4 and 5 is 3.81 mm (0.150 in) and between points 5 and 6 it is 2.54 mm (0.100 in). The flow through this top portion of the rib will not be represented correctly with Fusion triangular elements or with midplane triangular elements. Overall, the door panel can be easily represented, but in the grill area it cannot be modeled well.

If the grill area is a critical area, a 3D mesh could be used to represent entire part so the grill can be properly represented.

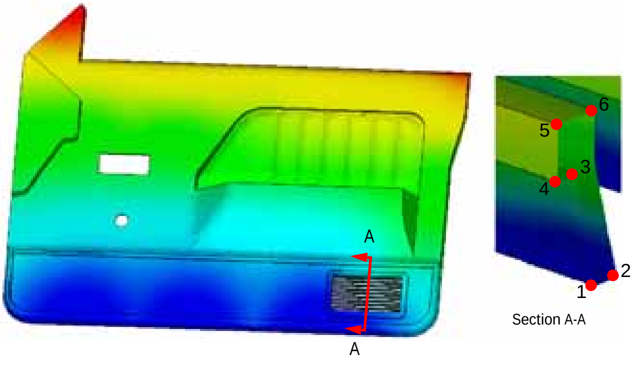


Figure 5.23 Door panel and grill cross section

5.5.2 Manifold

The manifold is a good example for using a 3D mesh. The part is very “chunky.” The thickness is not clearly defined on many areas for this part. It is not possible to create a midplane model that “looks good” for this part. And though a Fusion mesh would look good, it would not correctly represent the part because it would not properly define the part's thicknesses, and most of this part does not have a 4:1 width to thickness ratio.

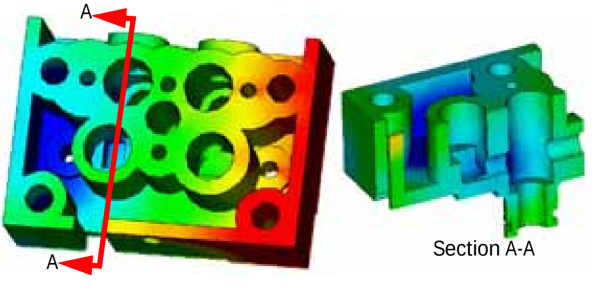


Figure 5.24 Manifold and cross section

6 Product Design

- Material properties for product design
- Design for strength
- Part thickness
- Boosting structural integrity with ribs
- Design for assembly

6.1 Material Properties for Product Design

6.1.1 Plastics Are Sensitive to Operating Conditions

The plastics molding processes allow parts designers more freedom than working with metals because plastics materials are so versatile. Unlike metals, however, the mechanical properties of plastics are very sensitive to the type, rate, duration, and frequency of loading; the change in operating temperature; and in some cases, relative humidity. The plastics part designer must take a material's response to these conditions into account. The table below lists the five typical loading and operating conditions, together with the relevant material properties a designer needs to consider.

Table 6.3: Typical loading/operating conditions and relevant material properties

Loading/operating conditions	Relevant material properties
Short-term loading	Stress-strain behavior
Long-term loading	Creep and stress relaxation
Repeated loading	Fatigue
High velocity and impact loading	Impact strength
Loading at extreme temperatures	Thermal mechanical behavior

6.1.2 Stress-Strain Behavior

6.1.2.1 Part Strength

The stress-strain behavior of a material determines the material contribution to part strength (or stiffness), which is the relationship between load and deflection in a plastic part. Other factors affecting part strength include part geometry, loading, constraint conditions on the part, and the residual stresses and orientations that result from the molding process. There are various types of strength, such as tensile, compressive, torsional, flexural, and shear, depending on the load and restraint conditions the part is subjected to. These types also correspond to the primary load state present in the part. The stress-strain behavior of the material in the same mode as the primary load state in the part is most relevant in determining part strength.

6.1.2.2 Tensile Properties

It is important to consider the relevant stress-strain behavior that corresponds to the primary (and, commonly, the multiple) load state at the operation temperature and strain rate. However, because of the inherent accuracy problems regarding the current testing procedures for nontensile tests, most of the published stress-strain data for plastics materials are limited to only short-term, load-to-failure tensile test results. Readers concerned about other types of load states than tensile properties should refer to other literature for relevant information.

Figure 6.1 depicts the tensile bar test sample and the deformation under a pre-set, constant load.

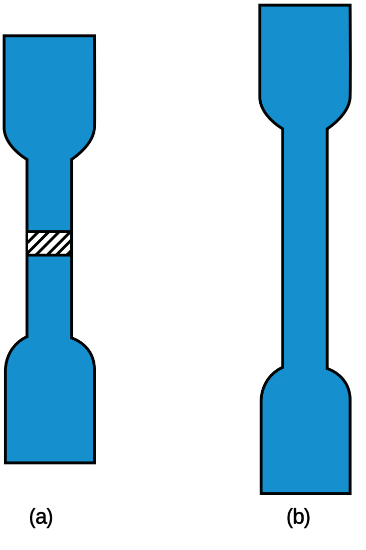


Figure 6.1 (a) Tensile test bar with a cross-sectional area, A, and original length, L_o
(b) Tensile test bar under a constant load, F, with elongated length, L.

The stress and strain are defined as:

$$\text{Stress } (\sigma) \equiv \frac{\text{Loading force (F)}}{\text{Cross-sectional area (A}} \tag{6.1}$$

$$\text{Strain } (\varepsilon) \equiv \frac{L - L_0}{L_0} \tag{6.2}$$

6.1.2.3 Viscoelastic Behavior and Spring / Dashpot Model

For viscoelastic materials, such as plastics, the short-term tensile test data tend to reflect values that are predominantly affected by the elastic response. However, you must also test and evaluate time-related viscoelastic behavior, as in the response to long-term loading, to determine any detrimental long-term effects. As one of the mathematical models, springs and dashpots in various combinations have been employed to model the response of plastics materials under load.

Springs Represent Elastic Response to Load: The spring in Figure 6.2 represents the elastic portion (usually short term) of a plastic material's response to load. When a load is applied to the spring, it instantly deforms by an amount proportional to the load. When the load is removed, the spring instantly recovers to its original dimensions. As with all elastic responses, this response is independent of time, and the deformation depends on the spring constant.

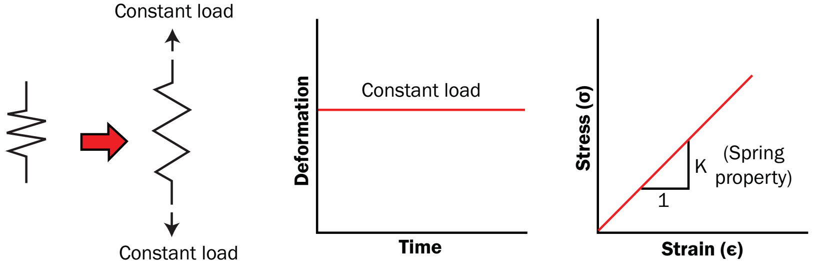


Figure 6.2 A spring represents the elastic response to load

Dashpots, which dampen movement to avoid shock, represent viscous response to load. The dashpot in Figure 6.3 represents the viscous portion of a plastic's response. The dashpot consists of a cylinder holding a piston immersed in a viscous fluid. The fit between the piston and cylinder is not tight. When a load is applied, the piston moves slowly in response. The higher the loading, the faster the piston moves. If the load is continued at the same level, the piston eventually bottoms out (representing failure of the part). The viscous response is generally time- and rate-dependent.

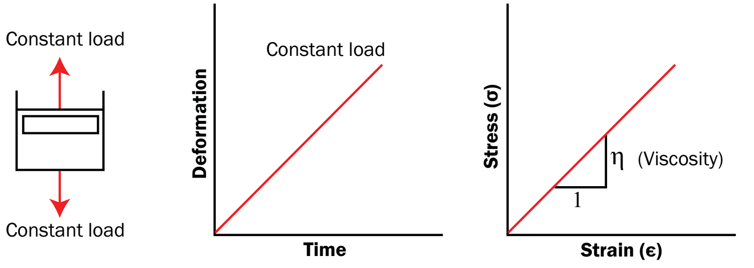


Figure 6.3 A dashpot represents a viscous response to load

Voight-Kelvin Mechanical Model Mimics Typical Response to Load: The Voight-Kelvin mechanical model, which includes a spring and dashpot in series with a spring and dashpot in parallel, is the most common model (see Figure 6.4) to mimic the plastics' behavior upon loading.

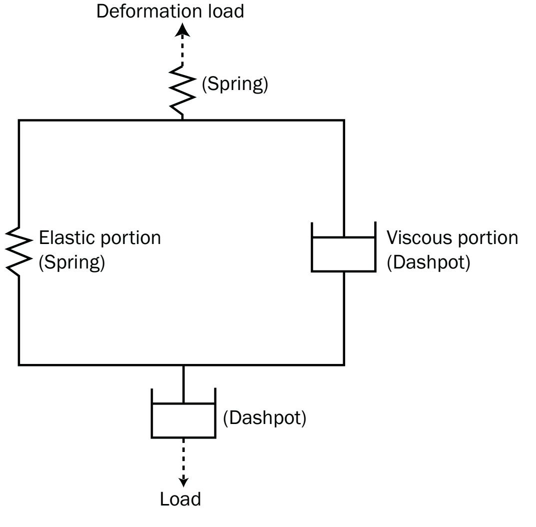


Figure 6.4 A Voight-Kelvin mechanical model mimics the typical behavior of a plastic's response to load

The components of the Voight-Kelvin mechanical model are:

- The spring in the series represents the elastic, recoverable response to a load.
- The dashpot in the represents a time-dependent response that may not be recoverable when the load is removed.
- The spring and dashpot in parallel represent a time-dependent response that is recoverable over time by the action of the elastic spring.

6.1.2.4 Stress-Strain Curves for Unfilled Polymers

Figure 6.5 shows a typical stress-strain curve for short-term loading of a typical unfilled thermoplastic material. Figure 6.6 depicts the same curve as shown in Figure 6.5 except it is stretched horizontally to show the details within the elastic region. Several important material properties, such as Young's modulus, proportional limit, elastic limit, yield point, ductility, ultimate strength, and elongation at failure, can be obtained from the stress-strain curve, as shown in Figure 6.5 and Figure 6.6.

Young's Modulus: Young's modulus is derived from the initial, straight-line portion of the curve as the ratio of stress to strain for that portion of the curve, shown in Figure 6.6.

$$\text{Young's modulus (E)} \equiv \frac{\sigma}{\varepsilon} \tag{6.3}$$

Although it is occasionally referenced as a measure of material strength, Young's modulus is actually more of an indicator of the rigidity of a material than the strength. It is the basis for simple linear engineering calculations, for example, in determining the stiffness of a plastic part.

Proportional Limit: The proportional limit, marked as point "P" in Figure 6.6, is the strain at which the slope of the stress-strain curve starts to deviate from linear behavior.

Elastic Limit: The elastic limit, point "I" on Figure 6.6, is the greatest strain the material can absorb and still recover. As strain continues to increase, the plastic will either draw, without recovery, or fail by rupturing (as shown in Figure 6.5).

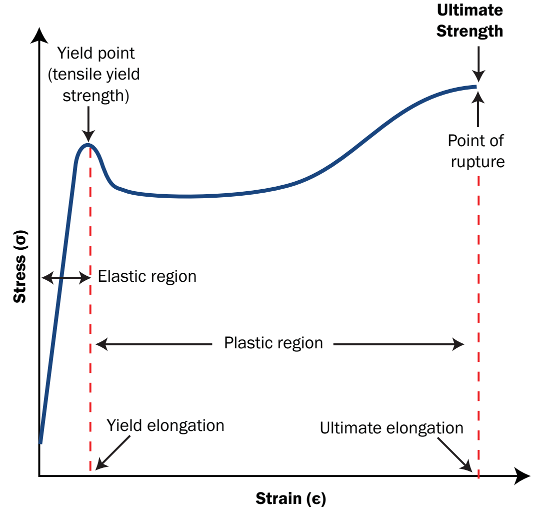


Figure 6.5 Stress-strain curve for a typical thermoplastic

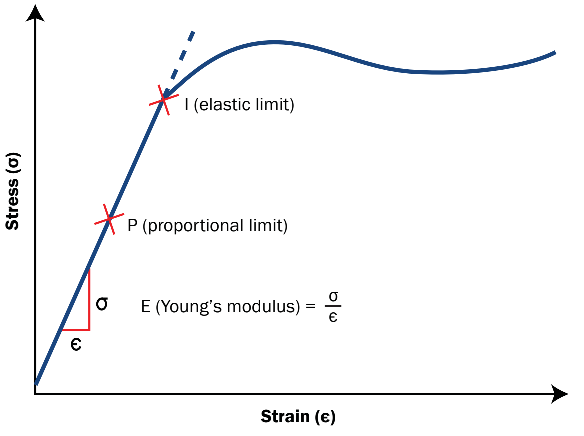


Figure 6.6 Details of the elastic region of the stress-strain curve shown in Figure 6.5: point P is the proportional limit, most often used as the design strain limit; point I is the elastic limit, beyond which the plastic part will not recover its original shape

6.1.2.5 Stress-Strain Curves for Fiber-filled polymers

The stress-strain curves for a pair of thermoplastic compounds are shown in Figure 6.7. The base resin is the same for both compounds, except one compound is unfilled while the other contains 30% glass fiber as reinforcement. You can see that the glass fibers significantly increase the ultimate strength, yield strength, proportional limit, and the Young's modulus while causing the filled resin to rupture at a much lower strain. On the other hand, the unfilled resin shows "drawing" at strains beyond the yield point. The stress decreases to a plateau beyond the yield point before failure. Typically, the cross-sectional area of the sample decreases during the drawing, according to Poisson's ratio for the material.

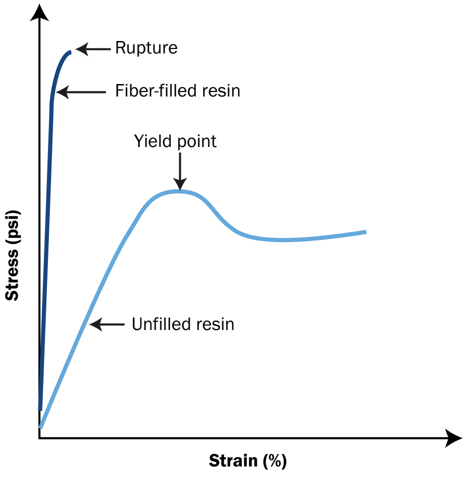


Figure 6.7 Stress-strain curves for a fiber-filled and an unfilled resin

6.1.2.6 Rate- and Temperature-dependency of Stress-Strain Curves

The loading rate (or the strain rate) and temperature can significantly affect the stress-strain behavior of plastics. As an illustration, Figure 6.8 plots the influence of loading rates and temperature on the tensile stress-strain curve for a semicrystalline resin. In general, at higher loading rates or lower temperatures, plastic materials appear to be more rigid and brittle. On the other hand, at lower loading rates or higher temperatures, materials appear to be more flexible or ductile because of their viscous characteristics. As you can see in Figure 6.8, an increase in loading rate significantly increases the ultimate and yield strength, whereas an increase in temperature leads to decreases in ultimate and yield strength and in proportional limit.

If the material is semicrystalline and the glass transition temperature is crossed when raising the temperature, these rate- and temperature-dependent effects can be very large, resulting in entirely different behaviors. If the material is amorphous and the softening range is crossed, the material will undergo viscous flow when loaded.

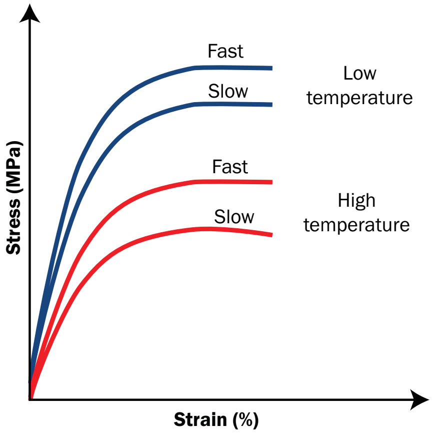


Figure 6.8 Stress-strain curves for a typical polymer at two test temperatures (high and low) and two rates of loading (fast and slow).

6.1.3 Creep and Stress Relaxation

Creep and stress relaxation are critical concerns when designing structural parts that are subject to long-term loading.

6.1.3.1 Creep

Regardless of the rate at which the initial load is applied, if a constant load is continued, the structure will continue to deform. This long-term, permanent deformation is called creep, as plotted in Figure 6.9.

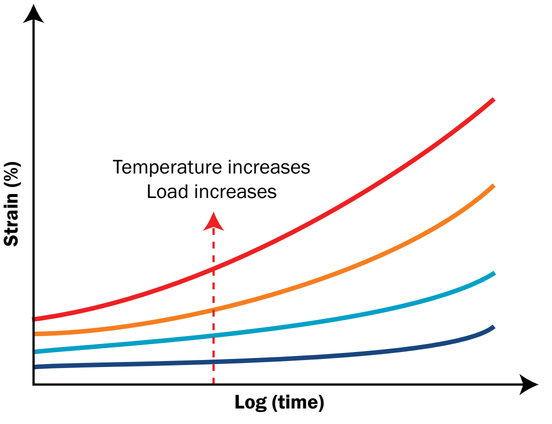


Figure 6.9 A typical creep in flexure curve; note that creep depends on load and time

To design parts subject to long-term loading, designers must use creep data to ensure that the parts do not rupture, yield, craze, or simply deform excessively over their service life. Although creep data exist for many resins at specific times, stress levels, and temperatures, each individual application must use the data that correlate with the type of stress and environmental conditions that the part is subjected to during service. Since the process of individual testing for long periods of time is not feasible and the stress and environmental conditions are difficult to predict over the long term, methods for interpolating and extrapolating shorter information are necessary. Engineers typically have to enter creep databases provided by resin suppliers to obtain time-strain data, then perform interpolation and extrapolation procedures to develop a complete nonlinear isochronous stress-strain curve, as shown in Figure 6.10. These curves are then used in place of short-term stress-strain curves when designing for applications involving long-term static loading.

6.1.3.2 Creep Modulus

The time- and temperature-dependent creep modulus, E_c, as a function of constant stress, σ, and time- and temperature-dependent strain, $\varepsilon(t,T)$, as defined below, can be used in design calculations for constant stress or strain-stress relaxation applications.

$$\text{Creep modulus } (E_c) = \frac{\sigma}{\varepsilon(t, T)} \tag{6.4}$$

Other factors associated with creep are:

- The rate of creep and stress relaxation will increase with increases in temperature.
- If the load is continued long enough, rupture may occur. This is called *stress cracking*.
- High internal (residual) stress should be considered along with the external stresses.

6.1.3.3 Stress Relaxation

Stress relaxation is a corollary phenomenon to creep. If the deformation is constant, the stress resisting that deformation will decrease with time. The physical mechanism that causes a plastic to undergo creep also applies to the phenomenon of stress relaxation. Figure 6.10 illustrates that at a fixed strain, the stress decreases with the elapsed time.

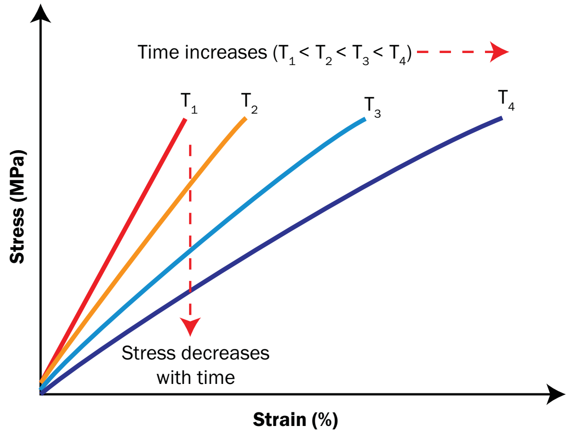


Figure 6.10 Isochronous (fixed-time) curves demonstrate stress relaxation at a constant strain (deflection)

6.1.4 Fatigue

Fatigue has to be considered when designing plastic parts that are subject to repeated loading. The cyclical loading application is relatively infrequent and there is a long time between applications. If the loading is cyclical, use the proportional limit for design calculations. If the loading is repeated at short intervals and for long periods, use the S-N (stress vs. number of cycles) curves as the design criterion.

6.1.4.1 S-N Curves

The S-N curves are obtained by tests run in bending, torsion, or tension at a given constant frequency, temperature, and amplitude of loading. The stress at which the plastic will fail in fatigue decreases with an increase in the number of cycles, as shown in Figure 6.11. With many materials, there is an endurance limit (corresponding to the stress at the level-off section) below which stress level fatigue failure is unlikely to occur.

6.1.4.2 Fatigue Phenomenon

Depending on the stress level, repeated loading to a relatively low stress level may not show complete recovery after each cycle. In addition, as the number of load and unload cycles increases, and the interval between loading decreases, microcracks on the surface or other physical defects could develop and over time lead to a decrease in overall toughness and eventual failure.

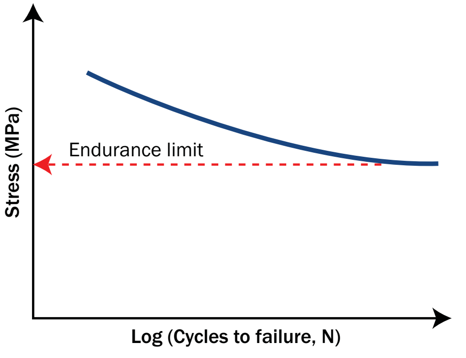


Figure 6.11 A typical flexural fatigue (S-N) curve with the endurance limit below which the repeated load is unlikely to cause fatigue

6.1.5 Impact strength

6.1.5.1 Toughness

Because plastics are viscoelastic, their properties strongly depend on the time, rate, frequency, and duration of the load, as well as the operating temperature. Impact strength (or toughness) of plastics can be defined as the ability of a material to withstand impulsive loading. Figure 6.8 shows that a material's impact strength increases with increasing rate of loading. The limit of this behavior is that as the velocity of loading increases, there is a reduced tendency to draw and the material acts in a brittle, rather than tough, fashion. Decreasing temperature shows a similar behavior, namely, at lower temperatures plastics are more brittle.

6.1.5.2 Stress Concentration

Impact response of plastic materials is also notch sensitive. In other words, a sharp internal radius will decrease the apparent impact strength of the part because of the effect of stress concentration, as plotted in Figure 6.12.

6.1.6 Thermal Mechanical Behavior

Changes in temperature can significantly change the dimension and mechanical performance of plastic parts. Therefore, you must consider both the high and low temperature extremes associated with the application. For applications subject to large temperature variation, you'll need to take into account the dimensional change of plastics parts when assembled/bound with other materials of different coefficient of thermal expansion (e.g., metals).

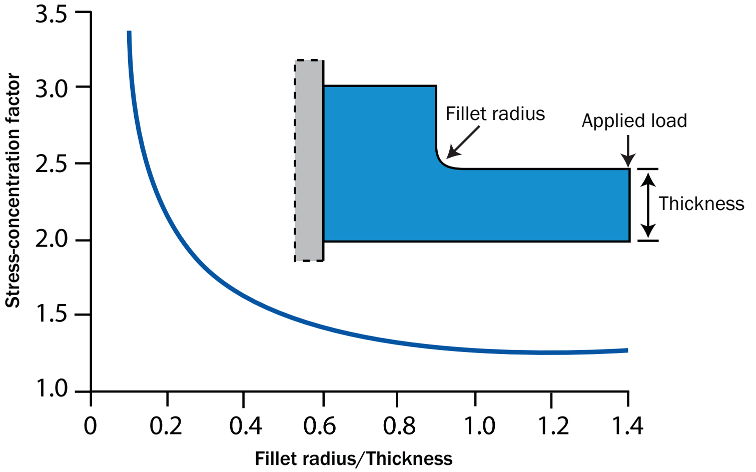


Figure 6.12 Stress concentration as a function of wall thickness and corner radius.

6.1.6.1 Operation at Extreme Temperatures

Factors to consider when the operating temperatures are above normal room temperature include:

- Part dimensions increase proportional to length, temperature increase, and coefficient of thermal expansion and contraction
- Strength and modulus will be lower than at room temperature; Figure 6.8 shows that strength decreases with increasing temperature
- Material may exhibit a rubber-like behavior with low modulus and high degree of drawing

6.1.6.2 Storage at Extreme Temperatures

Factors to consider for long-term storage at elevated temperatures include:

- Increased creep and stress relaxation for any components that are loaded during the storage; this includes relaxation of any residual stresses from the molding process or from assembly
- The plastic becomes brittle due to molecular degeneration
- Some of the ingredients bleed from the compound

Factors to consider when the storage temperatures are below room temperature include:

- Part dimensions decrease proportional to length, temperature decrease, and coefficient of thermal expansion and contraction
- Modulus increases
- Parts are more brittle

6.1.6.3 Coefficient of Thermal Expansion

The coefficient of thermal expansion measures the change in dimension from a specific temperature rise. The typical values (in the range of 10^{-4} 1/°K) are five to 10 times larger than those of metals. If the plastic part is rigidly joined to a metal part, the weaker plastic part will fail because of differential expansion or contraction. Depending on the strength of the plastic and the temperature rise, the failure may be immediate or delayed (see Section 6.1.2.6). The design must make allowances for the change in length between the plastic and the metal to which it is attached. If one end of the plastic is rigidly attached, the other end must be allowed to float.

The orientation of molecules and fibers might cause the change in dimension to be anisotropic.That is, the coefficient of thermal expansion (thus the expansion or contraction) is greater in one direction (e.g., the flow direction) than in the cross direction.

6.1.6.4 Heat Deflection Temperature Under Load

This value is derived from an ASTM test that includes soaking a standard test specimen in an oil bath of uniform temperature. A flexural load is applied after the specimen reaches the constant temperature of the oil bath. The temperature at which the specimen is deflected to a specified amount is called the heat deflection temperature. The test has little other meaning than to rank materials for heat resistance. Stress-strain curves for a range of temperatures provide a more reliable way of evaluating material's performance at elevated temperatures.

6.2 Design for Strength

6.2.1 Predicting Part Strength

The success or failure of the plastics product design is often determined by how accurately the part strength (stiffness) can be predicted. The types of strength correspond to the load and restraint conditions to which the part is subjected, such as tensile, compressive, torsional, flexural, and shear. The strength of a plastics part will depend on the material, the geometry of the part, constraint conditions on the part, and the residual stresses and orientations that result from the molding process.

6.2.2 Loading / Operating Conditions

The strength values that must be used for designing viable, long-lived plastics parts depend on the nature of the expected load:

- Short-term loading
- Long-term loading
- Repeated loading
- Enhance heat dissipation
- Loading at extreme temperatures

Relevant material properties associated with the various loading conditions are discussed in Section 6.1.1.

6.2.2.1 Short-term Loading

Short-term loads are those imposed during handling and assembly, and during usage where the load is applied occasionally with short durations. The following suggestions apply to parts that will be subject to short-term loading conditions.

Use Proportional Limit in Stress-Strain Curve: Designers should consider the stress-strain behavior of the plastic material when designing parts for bearing short-term loads. The proportional limit should be used as the maximum allowable stress in the design calculations to avoid permanent deformation of the part and possible loss of function.

Use Stiffeners and Fiber Reinforcements: Stiffeners, such as ribs and gussets (see Section 6.4), are often used to increase the part strength. Fiber reinforcements, oriented in a favorable direction, can also increase the part strength. Ribs should be considered for parts with large spans. Increasing the rib height and/or decreasing the span (spacing) between the ribs also improves part strength.

6.2.2.2 Long-term Loading

Long-term loading occurs when parts are placed under high external loads, within the proportional limit, for extended periods of time. This term also refers to parts that must withstand high internal or residual stresses that result from either the molding process or from the following assembly processes:

- Press-fit and snap-fit assemblies
- Tapered fit between plastic and metal components
- Over-stressed joints between mating parts
- Thread-forming screws
- Counter-bored screw heads

The design rules given below apply to parts that will be subject to long-term loading conditions.

Use Creep Modulus: Creep modulus should be used in the design calculations to avoid stress-cracking failure, to maintain the tightness of joints, and to maintain part functionality.

Designing for Press-fit and Snap-fit Assemblies: For press-fit joints and snap-fit joints, design snap-fit and press-fit components so that the strain is reduced to the as-molded dimensions after assembly.

Using Fasteners: There are several design alternatives you can use for incorporating fasteners into a plastics part. These strategies are discussed in Section 6.5.7.

Design Features to Avoid Over-tightening: Plastic-to-plastic surfaces should be designed to limit the distance that the joint can be closed. Providing stop surfaces can prevent a screw from being over-tightened beyond the design intent or limit the depth of engagement of two matching taper surfaces.

6.2.2.3 Repeated Loading

When parts are subject to conditions of repeated loading, the number of loads that part will be expected to withstand over its lifespan must be considered. The table below gives examples of types of repeated loads. The corresponding numbers are the expected number of times the loading may occur.

Table 6.4: Examples of repeated loads and expected number of loads.

Type of Load	Number of loads
Repeated assembly and disassembly	Less than 1,000
Gear teeth with rapidly repeated loading of each tooth	Greater than 10,000
Spring components	Greater than 10,000

The following suggestions apply if the part being designed will need to withstand repeated loadings, like the ones given above.

Cyclic Loadings: If the cyclical loading application is relatively infrequent and there is a long time between applications, use the proportional limit for design calculations.

Repeated Loadings: f the loading is repeated at short intervals and for long periods, use the S-N (stress vs. number of cycles) curves (see Section 6.1.4.1) as the design criterion.

Avoid Microcracks: Smooth surfaces, as produced by highly polished mold surfaces, reduce the tendency for microcracks to form.

Avoid Stress Concentration: To avoid stress concentration, use a smooth, generous radius in areas like corners where the width and thickness changes.

6.2.2.4 Enhance Heat Dissipation

At higher frequency or amplitudes with repeated loads, plastic parts tend to run hotter and fail sooner. Designing with thin walls and fatigue-resistant conductive materials is generally recommended to maximize heat transfer.

6.2.2.5 High-Velocity and Impact Loading

High-velocity loading refers to velocities greater than one meter per second, while impact loading refers to velocities greater than 50 meters per second. Avoid high-velocity and impact loading on areas that are highly stressed from residual and/or assembly stresses. When designing a part that must withstand these types of loading conditions, keep the following suggestions in mind.

Use Proportional Limit: Use the proportional limit in the design calculation for the expected loading rate range.

Avoid Stress Concentration: To avoid stress concentration, use a smooth, generous radius in areas like corners where the width and thickness change.

Avoid Material Degradation: High melt temperatures over a prolonged period of time can cause the resin to become brittle. The amount of time the resin is at high temperatures should be minimized by selecting a proper melt temperature and by sizing a proper injection barrel to fit the job.

6.2.2.6 Loading at Extreme Temperatures

Storage, shipping, and usage temperatures can easily exceed or go below the normal room temperature range of 20 to 30°C. Following are examples of conditions under which a part will need to withstand temperatures above or below the ambient room temperature.

Above Room Temperature: Plastics parts stored or operated in these conditions will need to accommodate very high temperatures, including

- Hot liquid containers
- Hot water plumbing components
- Devices containing heating elements
- Shipped in vehicles sitting in direct sunlight
- Stored in buildings without air conditioning

Below Room Temperature: Plastics parts stored or operated in these conditions will need to accommodate very low temperatures, including:

- Refrigeration components
- Shipped in the hold of an airplane

6.2.2.7 Designing for Extreme Temperatures

Parts need to be designed to accommodate the changes in temperature to which they will be exposed.

Use the Proportional Limit: Use the proportional limit for the expected exposure temperature in design calculations to avoid permanent distortion of the part.

Allow Differential Expansion and Contraction: Do not rigidly fasten materials with large differences in coefficient of thermal expansion. Use fastening methods that allow for the greater expansion and contraction of the plastics parts. Figure 6.18 gives recommendations for designing this type of plastic part. Alternatives include slots that allow the free end to expand on one axis while maintaining the location in the other two axes.

6.3 Part Thickness

6.3.1 Part Thickness Drives Quality and Cost

Many factors need to be taken into account when designing a part. These include functional and dimensional requirements, tolerance and assembly, artistic and esthetic appearance, manufacturing costs, environmental impacts, and post-service handling. Here we discuss the manufacturability of thermoplastic injection-molded parts considering the influence of part thickness on cycle time, shrinkage and warpage, and surface quality.

6.3.2 Cycle Time Increases with Thickness

Injection-molded plastic parts have to be cooled sufficiently before being ejected from the mold to avoid deformation from ejection. Parts with thick wall sections take longer to cool and require additional packing.

Theoretically, cooling time is proportional to the square of the heaviest part wall thickness or the power of 1.6 for circular features. Therefore, thick sections will prolong the press cycle, reducing the number of parts per unit time and increasing the cost per part.

6.3.3 Thick Parts Tend to Warp

Shrinkage is inherent in the injection-molding process. Excessive and nonuniform shrinkage, however, both globally and through the cross section of the part, will cause the part to warp. Warpage is a distortion where the surfaces of the molded part do not follow the intended shape of the design. The diagrams below illustrate how part thickness affects shrinkage and warpage.

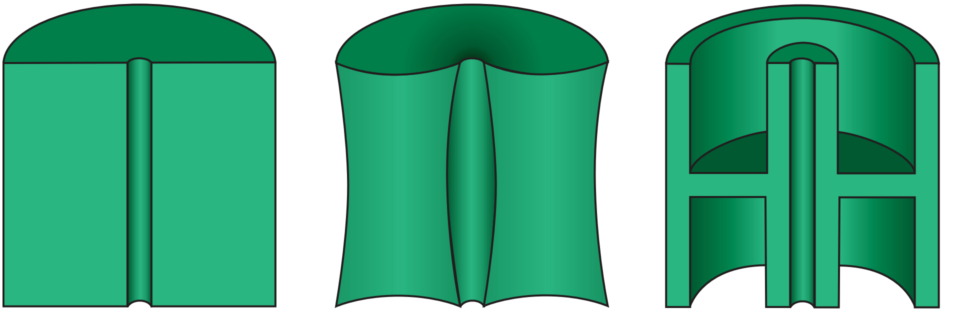


Figure 6.13 A thick part (left) can lead to excessive shrinkage and warpage (center); a recommended alternative design with uniform thickness (right)

6.3.4 Thin, Uniform Parts Improve Surface Quality

A combination of thin and heavy part cross sections can easily produce a racetracking effect, which occurs because melt preferentially flows faster along thick sections. Racetracking leads to air traps (see Section 12.1) and weld lines (see Section 12.17) that produce defects on part surfaces. In addition, sink marks and voids (see Section 12.16) will also arise in thick sections without sufficient packing.

6.3.5 Reducing Part Thickness

To shorten the cycle time, improve dimensional stability, and eliminate surface defects, a good rule of thumb for part thickness design is to keep part thickness as thin and uniform as possible. The use of ribs is an effective way to achieve rigidity and strength while avoiding heavy cross-sectional thickness.

Part dimensions should take into account the material properties of the plastics used in relation to the type of loading and operating conditions the part will be subjected to; the assembly requirements should also be considered.

6.4 Boosting Structural Integrity with Ribs

6.4.1 Structural Integrity: the Goal of Every Design

The major component of designing for structural integrity, in many cases, is to design the structure to be stiff enough to withstand expected loads. Increasing the thickness to achieve this is self-defeating, since it will:

- Increase part weight and cost proportional to the increase in thickness
- Increase molding cycle time required to cool the larger mass of material
- Increase the probability of sink marks

Well-designed ribs can overcome these disadvantages with only a marginal increase in part weight.

There are several common uses for ribs, including:

- Covers, cabinets, and body components with long, wide surfaces that must have good appearance with low weight
- Rollers and guides for paper handling where the surface must be cylindrical
- Gear bodies, where the design calls for wide bearing surfaces on the center shaft and on the gear teeth
- Frames and supports

6.4.2 Designing Ribs

Keep part thickness as thin and uniform as possible to shorten the cycle time, improve dimensional stability, and eliminate surface defects. The use of ribs is an effective way to achieve rigidity and strength, while avoiding heavy cross-sectional thickness. If greater stiffness is required, reduce the spacing between ribs, which enables you to add more ribs.

6.4.2.1 Rib Geometry

Rib thickness, height, and draft angle are related: excessive thickness will produce sinks on the opposite surface whereas small thickness and too great a draft will thin the rib tip too much for acceptable filling.

Ribs should be tapered (drafted) at one degree per side. Less draft can be used, to one half degree per side, if the steel that forms the sides of the rib is carefully polished. The draft will increase the rib thickness from the tip to the root, by about 0.175 mm per centimeter of rib height, for each degree of draft angle. The maximum recommended rib thickness, at the root, is 0.8 times the thickness of the base to which it is attached. The typical root thickness ranges from 0.5 to 0.8 times the base thickness. See Figure 6.14 for recommended design parameters.

6.4.2.2 Location of Ribs, Bosses, and Gussets

Ribs aligned in the direction of the mold opening are the least expensive design option to tool. As illustrated in Figure 6.14, a boss should not be placed next to a parallel wall; instead, offset the boss and use gussets to strengthen it. Gussets can be used to support bosses that are away from the walls. The same design rules that apply for ribs also apply for gussets.

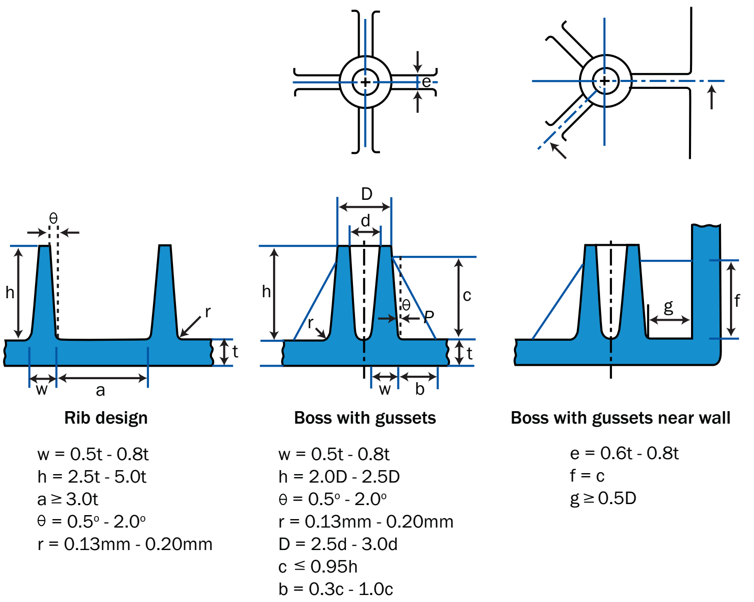


Figure 6.14 Recommendations for rib cross sections

6.4.2.3 Alternative Configurations

As shown in Figure 6.15, ribs can take the shape of corrugations. The advantage is that the wall thickness will be uniform and the draft angle can be placed on the opposite side of the mold, thereby avoiding the problem of the thinning rib tip.

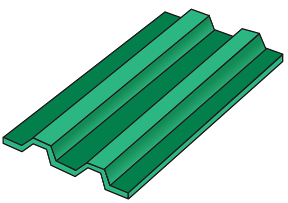


Figure 6.15 Corrugations instead of ribs

In terms of rigidity, a hexagonal array of interconnected ribs will be more effective than a square array, with the same volume of material in the ribs.

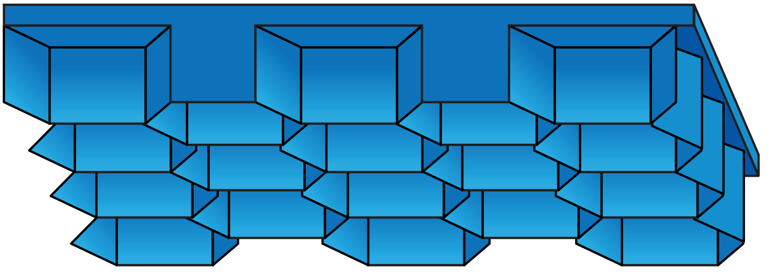


Figure 6.16 Honeycomb ribbing attached to a flat surface provides excellent resistance to bending in all directions

6.5 Design for Assembly

6.5.1 Molding One Part vs. Separate Components

A major advantage of molding plastics parts is that you can now mold what were previously several parts into one part. These include many of the functional components and many of the fasteners needed to assemble the molded part to other parts. Because of the limitations of the mold and the process, functional requirements, and/or economic considerations, however, it is still sometimes necessary to mold various components separately and then assemble them together.

6.5.2 Tolerances: Fit between Parts

Punched and machined parts can be made to tighter tolerances than molded parts because the large shrinkage from the melt to the solid state make sizing less predictable. In many cases, the solidification is not isotropic, so that a single value of mold shrinkage does not adequately describe the final dimensions of the parts.

6.5.2.1 Fit between Plastic Parts

- If the two plastic parts are made of the same material, refer to the tolerance capability chart supplied by the material supplier.
- If the two parts are of different material families or from different suppliers, add 0.001 mm/mm of length to the tolerances from the supplier's tolerance capability charts.

- If the flow orientations are strong, the isotropic shrinkages will require adding 0.001 mm/mm length to the overall tolerances of the parts.
- Add steps, off-sets, or ribs at the joint line of the two parts to act as interrupted tongue-and-groove elements to provide alignment of the two parts and ease the tolerance problem on long dimensions (see Figure 6.17).

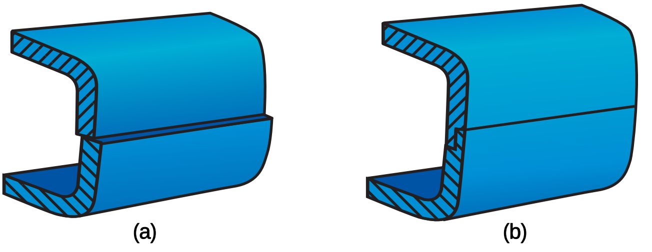


Figure 6.17 Butt joints (a) are difficult to align; matching half-tongue and groove (b) align the two parts within normal tolerances

6.5.2.2 Fit between Plastic Parts and Metal Parts

The joint between the plastic and metal must allow the plastic part to expand without regard to the expansion of the metal part.

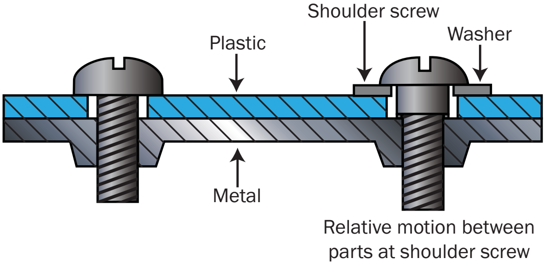


Figure 6.18 Design the joint between plastic and metal to allow for greater thermal expansion and contraction of the plastic; this includes using shouldered fasteners and clearance between the fastener and the plastic

6.5.3 Press-fit Joints

Simple interference fits can be used to hold parts together. The most common press-fit joint is a metal shaft pressed into a plastics hub. A design chart (e.g., Figure 6.19) recommended by the resin suppliers or interference formula can be used to design a press-fit joint at a desirable stress, so the parts will not crack because of excessive stress or loosen because of stress relaxation.

6.5.3.1 Interference Chart

Figure 6.19 plots the maximum interference limits as a percentage of the insert shaft diameter. Note that this chart is material specific and the maximum interference limit depends on the shaft material and the diameter ratio of the hub and insert. The maximum interference limit ($d-d_l$) as a percentage of the insert diameter, d, depends on the shaft material and the diameter ratio of the hub and insert (D/d). The recommended minimum length of interference is twice the insert diameter, 2d.

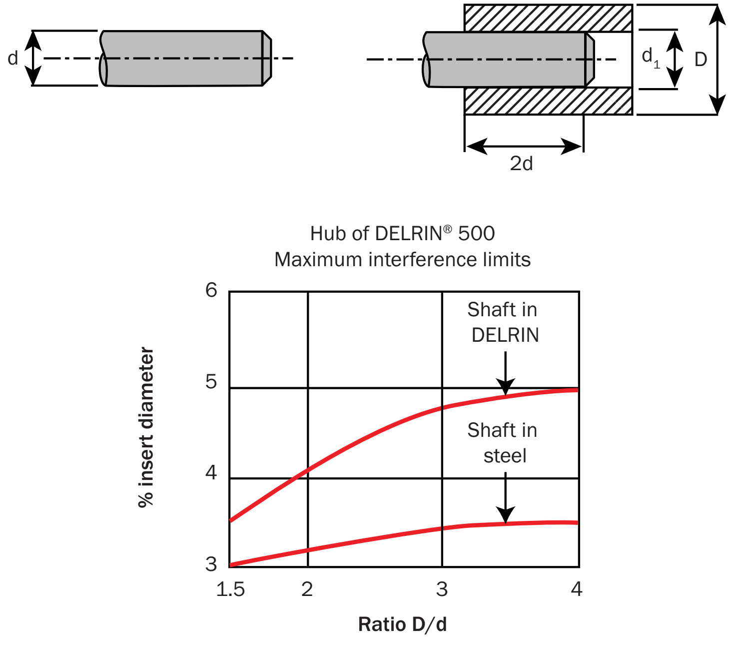


Figure 6.19 Maximum interference limits, pressing a metal shaft into a plastic hub; these curves are specific to the material

6.5.3.2 Interference Formula

If the relevant design chart is not available, the allowable interference (difference between the diameter of the insert shaft, d, and the inner diameter of the hub, d_1, see Figure 6.19) can be calculated with the following formula.

$$I = \left(\frac{S_d \times d}{W}\right) \times \left[\left(\frac{W + \nu_h}{E_h}\right) + \left(\frac{1 - \nu_s}{E_s}\right)\right] \tag{6.5}$$

$$W = \frac{1 + \left(\frac{d}{D}\right)^2}{1 - \left(\frac{d}{D}\right)^2}$$

where:

I = diametral interference ($d - d_l$), mm

S_d = design stress MPA

D = outside diameter of hub, mm

d = diameter of insert shaft, mm

E_h = tensile modulus of elasticity of hub, MPa

E_S = modulus of elasticity of shaft, MPa

V_h = Poisson's ratio of hub material

v_s = Poisson's ratio of shaft material

W = geometry factor

6.5.3.3 Tolerance

Check that tolerance build-up does not cause overstress during and after assembly and that the fit is still adequate after assembly.

6.5.3.4 Mating Metal and Plastic Parts

Do not design taper fits between metal and plastics parts, because stress cracking will occur from overtightening.

6.5.4 Snap-fit Joints

Snap-fit joints rely on the ability of a plastic part to be deformed within the proportional limit and returned to its original shape when assembly is complete. As the engagement of the parts continues, an undercut relieves the interference. At full engagement, there is no stress on either half of the joint. The maximum interference during assembly should not exceed the proportional limit. After assembly, the load on the components should only be sufficient to maintain the engagement of the parts.

Snap-fit joint designs include:

- Annular snap-fit joints
- Cantilever snap joints
- Torsion snap-fit joints

6.5.4.1 Annular Snap-fit Joints

Figure 6.20 illustrates a typical annular snap-fit joint. This is a convenient form of joint for axisymmetric parts. You can design the joint to be either detachable, difficult to disassemble, or inseparable, depending on the dimension of the insert and the return angle. The assembly force, W, strongly depends on the lead angle, α, and the undercut, y, half of which is on each side of the shaft. The diameter and thickness of the hub are d and t, respectively.

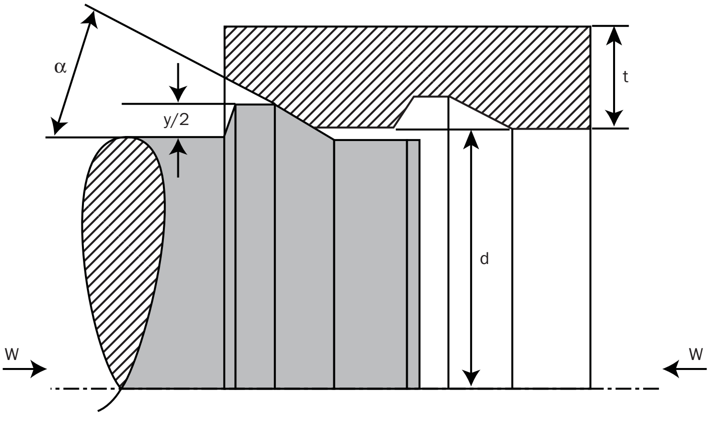


Figure 6.20 Typical annular snap fit joint

6.5.4.2 Hoop Stress

Figure 6.21 demonstrates that the outer member (assumed to be plastic) must expand to allow the rigid (usually metal) shaft to be inserted. The design should not cause the hoop stress, 6, to exceed the proportional limit of the material.

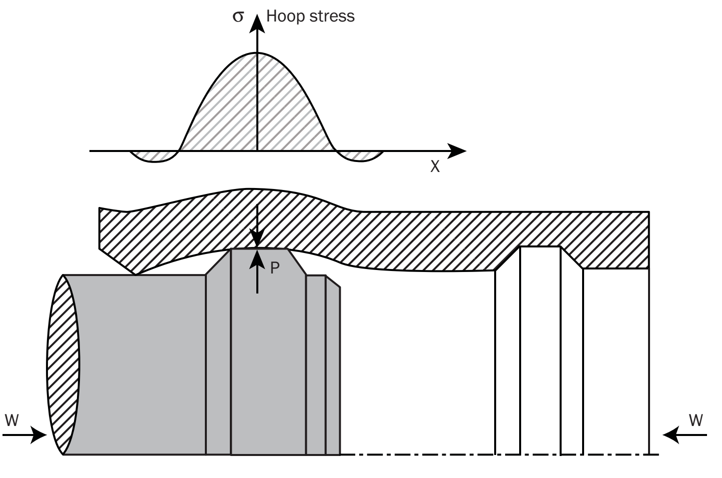


Figure 6.21 Stress distribution during the joining process

6.5.4.3 Permissible Deformation (Undercut)

The permissible deformation (or permissible undercut, y, shown in Figure 6.20) should not be exceeded during the ejection of the part from the mold or during the joining operation.

Maximum permissible strain: The maximum permissible deformation is limited by the maximum permissible strain, ε_{pm} and the hub diameter, *d*. The formula below is based on the assumption that one of the mating parts is rigid. If both components are equally flexible, the strain is half, i.e., the undercut can be twice as large.

$$y = c_{pm} \times d \tag{6.6}$$

Interference Ring: If the interference rings are formed on the mold core, the undercuts must have smooth radii and shallow lead angles to allow ejection without destroying the interference rings. The stress on the interference rings (see Equation 6.5) during ejection must be within the proportional limit of the material at the ejection temperature. The strength at the elevated temperature expected at ejection should be used.

6.5.5 Cantilever Snap Joints

This is the most widely used type of snap-fit joint. Typically, a hook is deflected as it is inserted into a hole or past a latch plate. As the hook passes the edge of the hole, the cantilever beam returns to its original shape. The beam should be tapered from the tip to the base, to more evenly distribute the stress along the length of the beam.

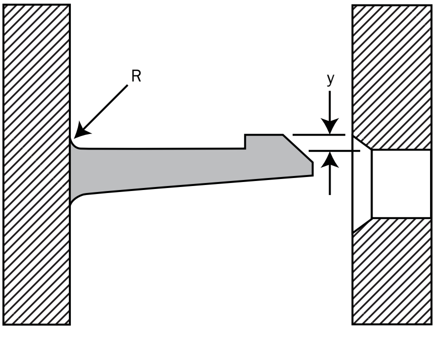


Figure 6.22 Typical cantilever snap-fit joint; the interference between the hole and the hook, y, represents the deflection of the beam as the hook is inserted into the hole

6.5.5.1 Proportional Limit

Assembly stress should not exceed the proportional limit of the material.

6.5.5.2 Designing the Hook

Either the width or thickness can be tapered (see Figure 6.22). Try reducing the thickness linearly from the base to the tip; the thickness at the hook end can be half the thickness at its base. Core pins through the base can be used to form the inside face of the hook. This will leave a hole in the base, but tooling will be simpler and engagement of the hook will be more positive.

6.5.5.3 Designing the Base

Include a generous radius on all sides of the base to prevent stress concentration.

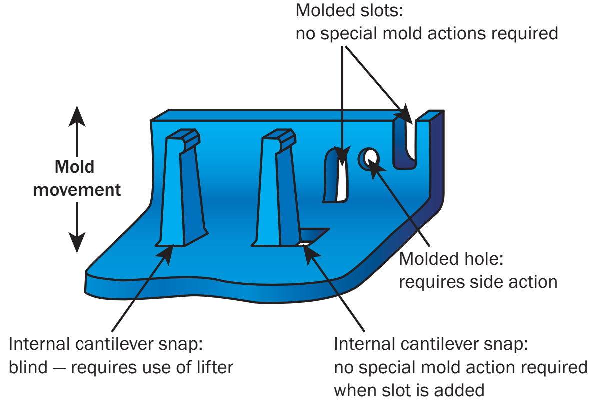


Figure 6.23 Design snap-fit features for ejection

6.5.6 Torsion Snap-fit Joints

In these joints the deflection is not the result of a flexural load as with cantilever snaps, but is due to a torsional deformation of the fulcrum. The torsion bar (see Figure 6.24) is subject to shear loads. This type of fastener is good for frequent assembly and disassembly.

6.5.6.1 Design Formula

The following relationship exists between the total angle of twist φ and the deflections y_1 or y_2:

$$\sin\varphi = \frac{y_1}{l_1} = \frac{y_2}{l_2} \tag{6.7}$$

where:

$$\varphi = \text{angle of twist}$$
$$y_1 \text{and } y_2 = \text{deflections}$$
$$l_1 \text{and } l_2 = \text{lengths of lever arms}$$

The maximum permissible angle φ_{pm} is limited by the permissible shear strain γ_{pm}:

$$\varphi_{pm} = \frac{180}{\pi} \times \gamma_{pm} \times \frac{l}{r} \tag{6.8}$$

where:

φ_{pm} = permissible total angle of twist in degrees
γ_{pm} = permissible shear strain
l = length of torsion bar
r = radius of torsion bar

The maximum permissible shear strain γ_{pm} for plastics is approximately equal to:

$$\begin{aligned} \gamma_{pm} &= (1 + v)\varepsilon_{pm} \\ \gamma_{pm} &\approx 1.35\varepsilon_{pm} \end{aligned} \tag{6.9}$$

where:

γ_{pm} = permissible shear strain
ε_{pm} = permissible strain
v = Poissons ratio (approx. 0.35 for plastics)

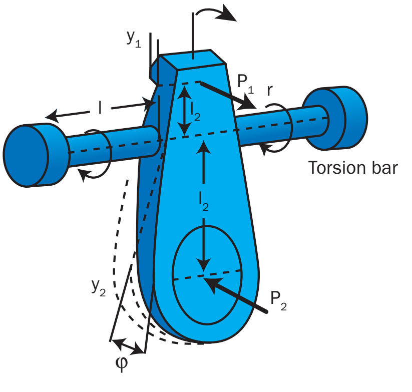


Figure 6.24 Torsional snap fitting arm with torsional bar

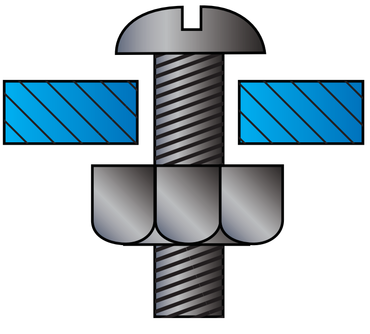
6.5.7 Fasteners

Screws and rivets, the traditional methods of fastening metal parts, can also be used with plastics. Several important concerns are:

- Over-tightening the screw or rivet could result in induced stress.
- Threads might form or be cut as the screw is inserted.
- Burrs on the screwhead or nut or on the head of the rivet could act as stress risers and cause early failure.

6.5.7.1 Screws and Rivets

Use smooth pan-head screws with generous pads for the head. Washers under the screw or rivet head should be burr-free or the punch-face should be against the plastic (die-face will have burrs from the stamping process). Figure 6.25 provides recommendations for the diameter of clearance holes for various screw sizes.



Screw SIze	Hole Diameter (in)	Hole Diameter (mm)
#2	.096	2.44
#4	.122	3.10
#6	.148	3.76
#8	.174	4.42
#10	.200	5.08
#12	.226	5.74
1 /4	.260	6.60
5/16	.323	8.20
3/8	.385	9.78

Figure 6.25 Recommendations for clearance between the machine screw and hole in the plastic; the panhead style of screw is recommended

Table 6.5: Recomended uses of various fasteners

Use	If
Thread-forming screws: ASA Type BF	The modulus of the plastics is less than 200,000 psi
Thread-cutting screws: ASA Type T, (Type 23) or Type BT (Type 25)	The modulus is greater than 200,000 psi, since thread-forming screws can cause stress cracking in this case
A metal, threaded cap with one screw thread on the boss.	The screw is to be removed and replaced many times. This will assure that later insertions do not cut or form a new thread or destroy the old one
Counter-bore hole with pan-head screw	The screw head must be below the surface of the part
Rivets to join plastic parts for a permanent assembly	The design prevents over-tightening of the joint or washers are used to prevent the head from cutting into the plastic

Table 6.6: Recommended fasteners to avoid

Do not use	Because
Countersunk screw heads	They are easily over-tightened and cause stress-cracking
Pipe threads	The tapered nature of this thread style can allow the joint to be easily overtightened and overstressed; stress-cracking will result

6.5.7.2 Molded Threads

Molding threads into the plastic component avoids having to use separate fasteners, such as screws and rivets. With molded threads tool-making will be easier if you provide a lead-in diameter slightly larger than the main diameter and about one screw flight long. Figure 6.26 shows how to design an unthreaded lead-in.

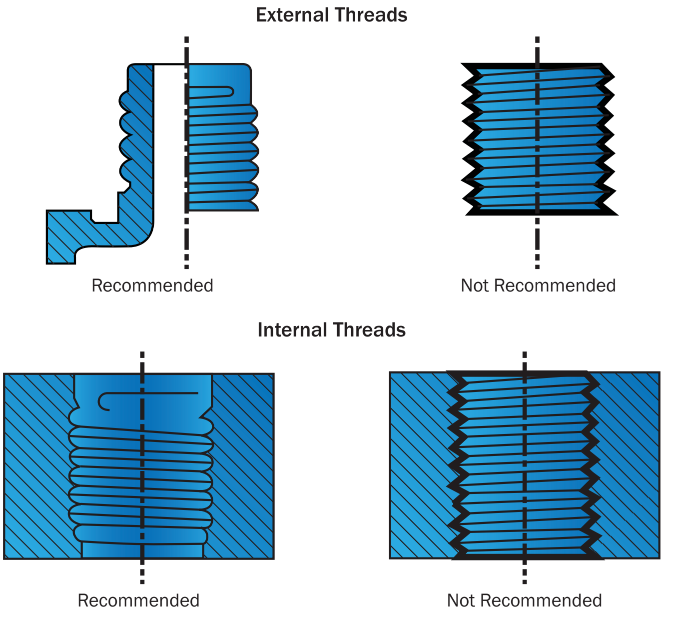


Figure 6.26 Recommended design for molded threads

Table 6.7: Design rules for molded threads

Task	Design rule
Thread size	Threads should be strong enough to meet the expected loads. Threads that are too small, especially if they're mated with metal threads, tend to become deformed and lose their holding power.
Inside radius of the thread	The thread design should avoid sharp inside radii. The corollary is that the peak of the thread should also be rounded to ease tool making.
Orienting threads to the parting line	If the axis of the thread is parallel to the mold parting line, half of the diameter can be molded in each mold half. You can reduce the effects of the parting line mismatch by partially flattening the threads at that point. Retractable mold components must be used if the axis of the threads is not parallel to the parting line.
Demolding the threads	Internal threads usually require unscrewing the mold component from the part, either manually or by action of the mold. Large internal threads can be formed on collapsing mold components.

6.5.8 Inserts

An insert is a part that is inserted into the cavity and molded into the plastic. The insert can be any material that will not melt when the plastic is introduced into the cavity. Metal inserts are used for electrical conductivity, to reinforce the plastic, and to provide metal threads for assembly. Plastics inserts can provide a different color or different properties to the combinations.

6.5.8.1 Balancing Melt Flow

Place the gate so that equal melt flow forces are placed on opposing sides of the insert. This will keep the insert from moving or deforming during mold filling. Design adequate flow paths so that the melt front proceeds at the same rate on either side of the insert.

6.5.8.2 Support Posts

Design support posts into the mold (these will be holes in the part) to support the insert.

6.5.8.3 Shrinkage and Weld Lines

Allow for shrinkage stress and for the weld line that will typically form on one side of the boss around the insert.

6.5.9 Welding Processes

Ultrasonic welding uses high-frequency sound vibrations to cause two plastics parts to slide against each other. The high-speed, short-stroke sliding between the two surfaces causes melting at the interface. When the vibrations are stopped, the melted interface cools, bonding the two surfaces. Other welding processes are generally not reliable or involve considerable hand work.

6.5.9.1 Design Rules for Welding

- The two materials must be melt compatible.
- The design of the ultrasonic horn that transfers energy to one of the plastics parts is important to success.
- Design axis-symmetrical parts with an interference at the joint. This is melted and the parts are forced together.
- The design of the contact surfaces is critical to success. An energy director, a small triangular raised bead, must be designed on one of the faces to be welded.

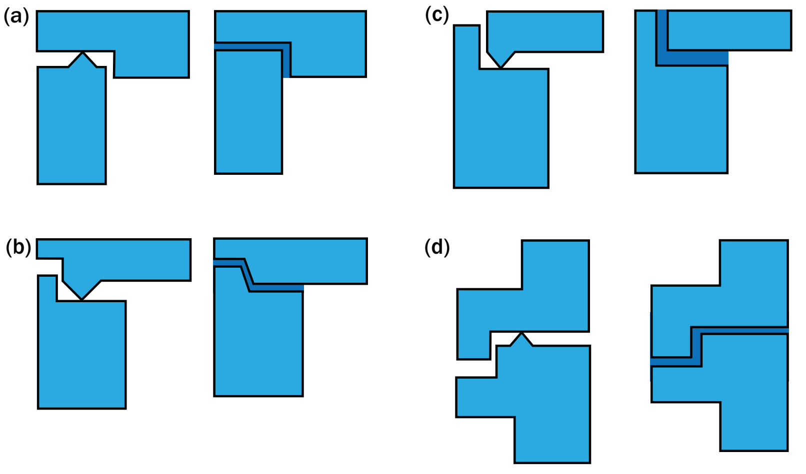


Figure 6.27 Recommendations for the design of ultrasonic welded joints

7 Gate Design

- Gate design overview
- Gate types
- Design rules
- Using Moldflow for gate design

7.1 Gate Design Overview

7.1.1 What Is a Gate?

A gate is a small opening (or orifice) through which the polymer melt enters the cavity. Gate design for a particular application includes selection of the gate type, dimensions, and location. It is dictated by the part and mold design, the part specifications (e.g., appearance, tolerance, concentricity), the type of material being molded, the fillers, the type of mold plates, and economic factors (e.g., tooling cost, cycle time, allowable scrap volume). Gate design is of great importance to part quality and productivity.

7.1.2 Single vs. Multiple Gates

How many gates to use is discussed in Section 4.2.1.1. Normally one gate is best, but additional gates may be needed to reduce the fill pressure or achieve a desired filling pattern.

7.1.3 Gate Dimensions

The cross section of the gate is typically smaller than that of the part runner and the part, so that the part can easily be “de-gated” (separated from the runner) without leaving a visible scar on the part. The gate thickness is nominally two-thirds the part thickness; however, the final size of the gate should be based on the shear rate in the gate and the application of the part. Since the end of packing can be identified as the time when the material in the gate drops below the freeze temperature, the gate thickness controls the packing time. A larger gate will reduce viscous (frictional) heating, permit lower velocities, and allow the application of higher packing pressure for a longer period of time. Choose a larger gate if you're aiming for appearance, low residual stress, and better dimensional stability. Figure 7.1 below illustrates the terms we use to describe gate a typical edge gate.

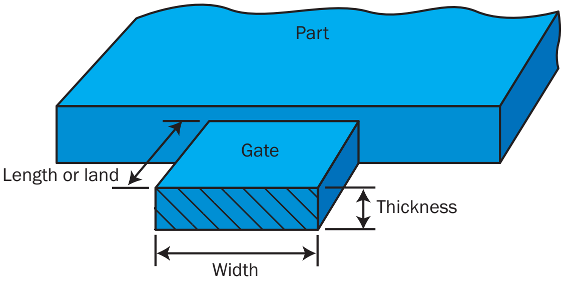


Figure 7.1 Gate size terminology

7.1.4 Gate Location

Many parameters must be considered when determining the gate location. In Section 4.2.1.2 the fundamental issues related to gate location are discussed from a design principles point of view. Section 7.3 discusses additional considerations for placing the gate. Gates are often placed in less than optimal part areas because of part usage and tooling related concerns. Moldflow can help evaluate many possible gate locations to determine the best location within design restrictions.

7.2 Gate Types

Gates can have a variety of configurations. They are classified into two categories based on the method of de-gating:

- Manually trimmed
- Automatically trimmed

7.2.1 Manually Trimmed Gates

Manually trimmed gates are those that require an operator to separate parts from runners during a secondary operation. The reasons for using manually trimmed gates are:

- The gate is too bulky to be sheared from the part as the tool is opened
- Some shear-sensitive materials (e.g., PVC) should not be exposed to the high shear rates inherent to the design of automatically trimmed gates
- Simultaneous flow distribution across a wide front to achieve specific orientation of fibers or molecules often precludes automatic gate trimming

Direct (sprue), tab, edge (standard), overlap, fan, disk (diaphragm), ring, spoke (spider), and film (flash) gate types are trimmed from the cavity manually.

7.2.1.1 Direct (Sprue) Gate

A direct (or sprue) gate is commonly used for single cavity molds, where the sprue feeds material directly into the cavity rapidly with minimum pressure drop. The disadvantage of using this type of gate is the gate mark left on the part surface after the runner (or sprue) is trimmed off. Freeze-off is controlled by the part thickness rather than determined the gate thickness. Typically, the part shrinkage near the sprue gate will be low, and shrinkage in the sprue gate will be high. This results in high tensile stresses near the gate.

Dimensions: The starting sprue diameter is controlled by the machine nozzle. The sprue's orifice diameter must be about 1.0 mm (1/32 in) larger than the nozzle exit diameter. Standard sprue bushings have a taper of between 1 and 3 degrees included angle, getting larger than the orifice diameter. Therefore, the sprue length will control the diameter of the sprue where it meets the part (the gate). The diameter of the gate is generally much greater than the thickness of the part. Nonstandard sprues, however, can be used:

- A smaller taper angle (a minimum of one degree) risks not releasing the sprue from the sprue bushing on ejection
- A larger taper wastes material and extends cooling time
- Nonstandard sprue tapers will be more expensive with little gain

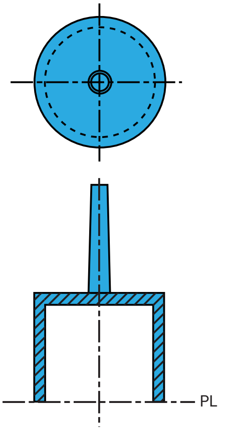


Figure 7.2 Sprue gate

7.2.1.2 Tab Gate

A tab gate typically is employed for flat and thin parts to reduce the shear stress in the cavity. The high shear stress generated around the gate is confined to the auxiliary tab, which is trimmed off after molding. A tab gate is used extensively for molding PC, acrylic, SAN, and ABS types of materials.

Dimensions: The minimum tab width is 6.4 mm. The minimum tab thickness is 75% of the depth of the cavity.

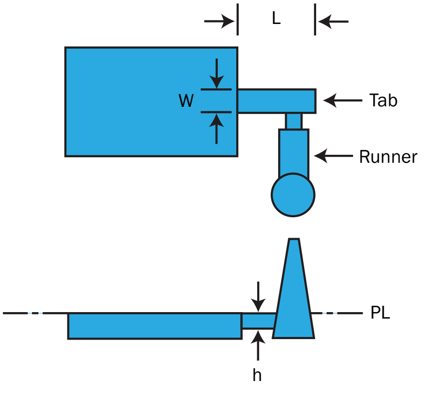


Figure 7.3 Tab gate

7.2.1.3 Edge (Standard) Gate

An edge gate is located on the parting line of the mold and typically fills the part from the side, top, or bottom.

Dimensions: The typical gate size is six to 75% of the part thickness (or 0.4 to 6.4 mm thick) and 1.6 to 12.7 mm wide. The gate land should be no more than 1.0 mm in length, with 0.5 mm being the optimum.

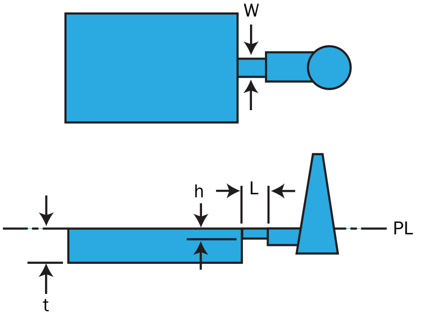


Figure 7.4 Edge gate

7.2.1.4 Overlap Gate

An overlap gate is similar to an edge gate, except the gate overlaps the wall or surfaces. This type of gate is typically used to eliminate jetting.

Dimensions: The typical gate size is 0.4 to 6.4 mm thick and 1.6 to 12.7 mm wide.

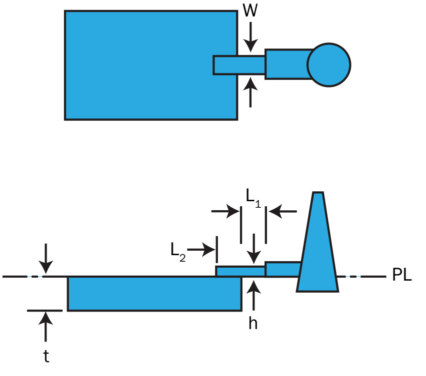


Figure 7.5 Overlap gate

7.2.1.5 Fan Gate

A fan gate is a wide edge gate with variable thickness. It permits rapid filling of large parts or fragile mold sections through a large entry area. It is used to create a uniform flow front into wide parts, where warpage and dimensional stability are main concerns.

The gate should taper in both width and thickness, so the flow front at the end of the gate is uniform. This will ensure that:

- The melt velocity will be constant at the end of the gate.
- The entire width is being used for the flow.
- The pressure is the same across the entire width.

Dimensions: The gate land is a narrow portion of the gate just before it enters the part. Typically, this will be a uniform cross section. The body of the gate is the balanced portion to achieve the balanced nature of the gate. The land thickness can be very thin relative to the part thickness because the gate is very wide. The gate shear rate will still be low. A thickness of ~1.0 mm gate land is common. The width of the fan gate is typically 25 mm and higher. Fan gates over 750 mm wide are used on very large parts. Often fan gates are as wide as the part itself.

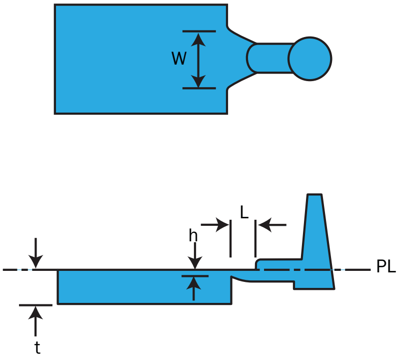


Figure 7.6 Fan gate

7.2.1.6 Disk (Diaphragm) Gate

A diaphragm gate is often used for gating cylindrical or round parts that have an open inside diameter. It is used when concentricity is an important dimensional requirement and the presence of a weld line is objectionable. This gate is essentially a flash gate around the inside edge of the part. Since the diaphragm is fed from a concentric sprue (or stub-runner drop), uniform flow to all parts of the gate is easy to maintain.

Dimensions: The typical gate thickness is 0.25 to 1.27 mm.

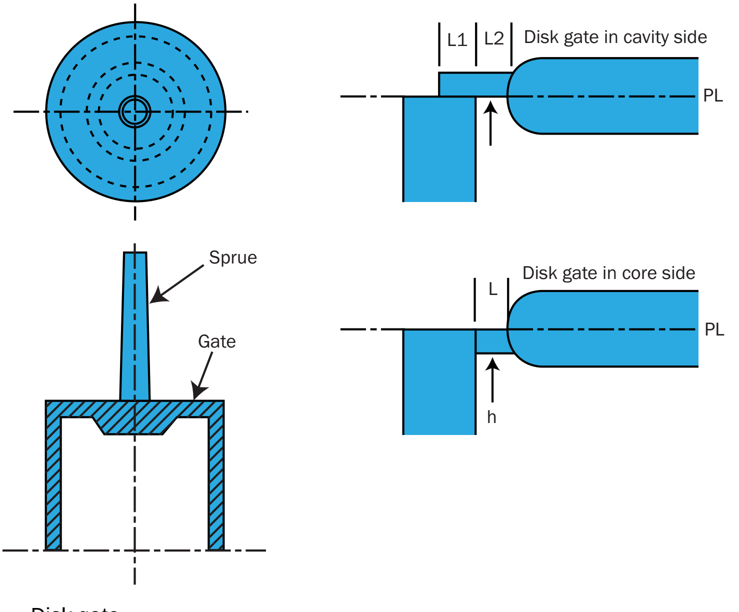


Figure 7.7 Disk gate

7.2.1.7 Ring Gate

Like a diaphragm gate, a ring gate is also used for cylindrical or round parts, but it is not always recommended. With a ring gate, the material flows freely around the core before it moves down as a uniform tube-like extrusion to fill the mold.

Dimensions: The typical gate thickness is 0.25 to 1.6 mm.

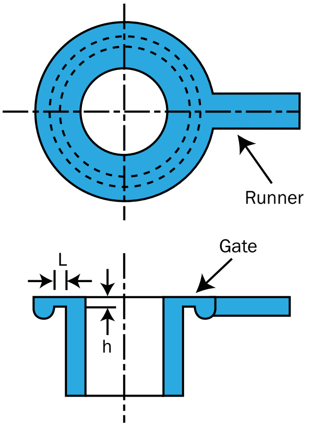


Figure 7.8 Ring gate

7.2.1.8 Spoke (Spider) Gate

This kind of gate is also called a four-point gate or cross gate. It is used for tube-shaped parts and offers easy de-gating and material savings. Disadvantages are weld lines and perfect roundness is unlikely.

Dimensions: Typical gate size ranges from 0.8 to 4.8 mm thick and 1.6 to 6.4 mm wide.

7.2.1.9 Film (Flash) Gate

A film gate is similar to a ring gate, but it is used for straight edges. It consists of a straight runner and a gate land across either the entire length or width of the cavity or a portion of the cavity. It is used for acrylic parts, and generally for flat designs of large areas where warpage must be kept to a minimum. This is a poor version of a fan gate. This gate is not likely going to have a flat flow front. There will not be uniform flow from the gate.

Dimensions: The gate size is small, approximately 0.25 to 0.63 mm thick. The land area (gate length) must also be kept small, approximately 0.63 mm long.

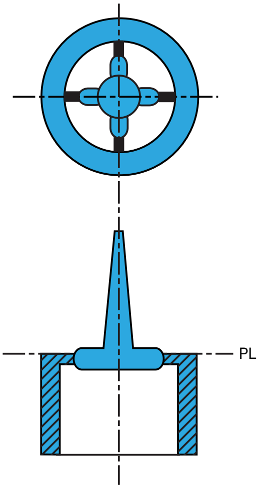


Figure 7.9 Spoke gate

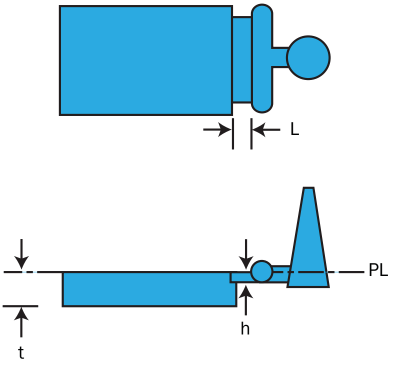


Figure 7.10 Film gate

7.2.2 Automatically Trimmed Gates

Automatically trimmed gates incorporate features in the tool to break or shear the gate as the molding tool is opened to eject the part. Automatically trimmed gates should be used to:

- Avoid gate removal as a secondary operation
- Maintain consistent cycle times for all shots
- Minimize gate scars

Pin, submarine (tunnel, chisel), hot-runner (hot-probe), and valve gate types are trimmed from the cavity automatically.

7.2.2.1 Pin Gate

This type of gate relies on a three-plate mold design, where the runner system is on one mold parting line and the part cavity is in the primary parting line. Reverse taper runners drop through the middle (third) plate, parallel to the direction of the mold opening. As the mold cavity parting line is opened, the small-diameter pin gate is torn from the part. A secondary opening of the runner parting line ejects the runners. Alternatively, the runner parting line opens first. An auxiliary, top-half ejector system extracts the runners from the reverse taper drops, tearing the runners from the parts.

Dimensions: Typical gate sizes are 0.25 to 1.6 mm in diameter. The shear rate from this type of gate will always be above the recommendations for the material.

Benefits: The design is particularly useful when multiple gates per part are needed to assure symmetric filling or where long flow paths must be reduced to assure packing to all areas of the part.

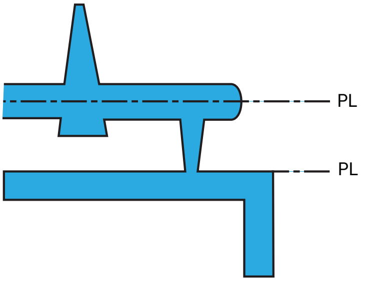


Figure 7.11 Pin gate

7.2.2.2 Submarine (Tunnel, Chisel) Gate

A submarine (sub) gate is used in two-plate mold construction. An angled, tapered tunnel is machined from the end of the runner to the cavity, just below the parting line. As the parts and runners are ejected, the gate is sheared at the part.

If a large diameter pin is added to a non-functional area of the part, the submarine gate can be built into the pin, avoiding the need of a vertical surface for the gate. If the pin is on a surface that is hidden, it does not have to be removed.

Multiple submarine gates into the interior walls of cylindrical parts can replace a diaphragm gate and allow automatic de-gating. The out-of-round characteristics are not as good as those from a diaphragm gate, but are often acceptable.

Dimensions: The typical size is 0.25 to 2.0 mm in diameter. It is tapered to the spherical side of the runner.

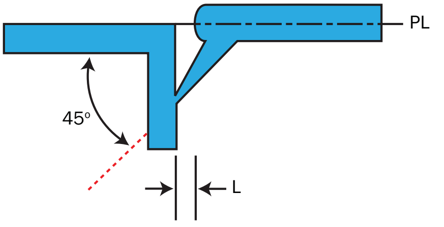


Figure 7.12 Submarine gate

7.2.2.3 Hot-runner (Hot-probe) Gate

A hot-runner gate is generally used to deliver hot material through heated runners and electrically heated sprues directly into the cavity. This is sometimes called runnerless moldings. The actual gate geometry can vary widely depending on the manufacturer and style of the gate.

The packing cycle is controlled by the freeze-off of the part near the gate. The very hot material at the gate is torn from the part as the cavity is opened.

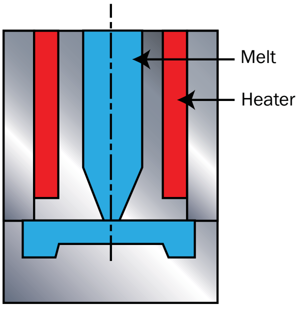


Figure 7.13 Hot runner

7.2.2.4 Valve Gate

The valve gate adds a valve rod to the hot runner gate. The valve can be activated to close the gate just before the material near the gate freezes. This allows a larger gate diameter and smoothes over the gate scar. Since the packing cycle is controlled by the valve rod, better control of the packing cycle is maintained with more consistent quality.

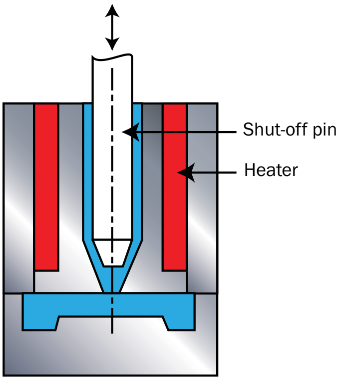


Figure 7.14 Valve gate

7.3 Design Rules

The design rules for gates is largely based on the concepts described in Chapter 4, in particular the Moldflow flow concepts including:

- Section 4.3.1, Unidirectional and Controlled Flow Pattern
- Section 4.3.2, Flow Balancing
- Section 4.3.6, Positioning Weld and Meld Lines
- Section 4.3.7, Avoid Hesitation Effects
- Section 4.3.8, Avoid Underflow
- Section 4.3.9, Balancing with Flow Leaders and Flow Deflectors

Below we will be expanding concepts introduced in Chapter 4, including:

- Determining the number of gates
- Flow patterns
- Positioning the gate

7.3.1 Determining the Number of Gates

The number of gates required for the part is primarily determined by the pressure requirements. Considering just the part (no runners) the maximum pressure required to fill should be no higher than about half the machine capacity.

In addition to pressure, gates may need to be added to achieve a balanced filling pattern. Figure 7.15 shows an example of a long narrow box. With one center gate the pressure drop along path P_1 is significantly more than the pressure along path P_2. With this single gate location, the center of the part will be overpacked, leading to warpage and a heavier part than is desired.

Figure 7.16 shows the same part with two gates. Now the flow lengths of paths P_1 and P_2 are the same so the pressure required to fill is the same. Each time a gate is added, the area of the part filled by a gate can be thought of a sub-molding. When breaking up a part into sub-moldings the following principles should be followed:

- Equal pressure drop in each sub-molding
- Equal volume in each sub-molding
- Position the weld/meld lines in the least sensitive areas
- Avoid hesitation effects
- Avoid underflow effects

Rarely can you completely satisfy all the these principles. Generally some compromises among these principles need to be made.

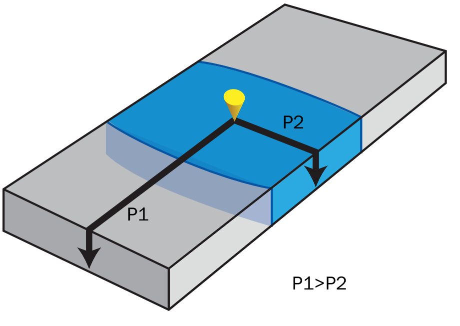


Figure 7.15 Gate location does not produce a balanced filling pattern

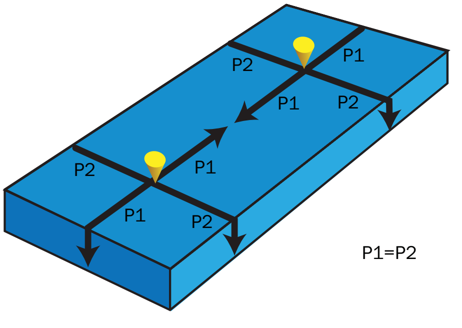


Figure 7.16 Gate locations produce a balanced filling pattern

7.3.2 Flow Patterns

The filling patterns for your parts should be as smooth and uniform as possible. The polymer will fill the mold with a straight flow front, without changing direction throughout the filling phase. Often on multigated parts this requires a runner system that is balanced so each gate is fed the correct volume of material at the pressures used for molding.

Figure 7.17 (a) shows a fan gate on the end of the part. The flow pattern is very balanced and unidirectional and normally will produce very little warpage. Figure 7.17 (b) is the same part as (a), but this time the gate locations are placed on the bottom edge of the part. In this case the flow front starts with a radial component, but quickly turns into a unidirectional flow front. The balance for this gating design is good, with the warpage being a little higher than the fan gate. Figure 7.17 (c) shows the part with four hot gates along the center axis. For this part, there is considerable radial flow, underflow, and overpacking in the center of the part, all contributing to a poorer design. Here the warpage is significantly worse than with the other gating locations.

Generally, the more unidirectional the flow front is, the less warpage there will be.

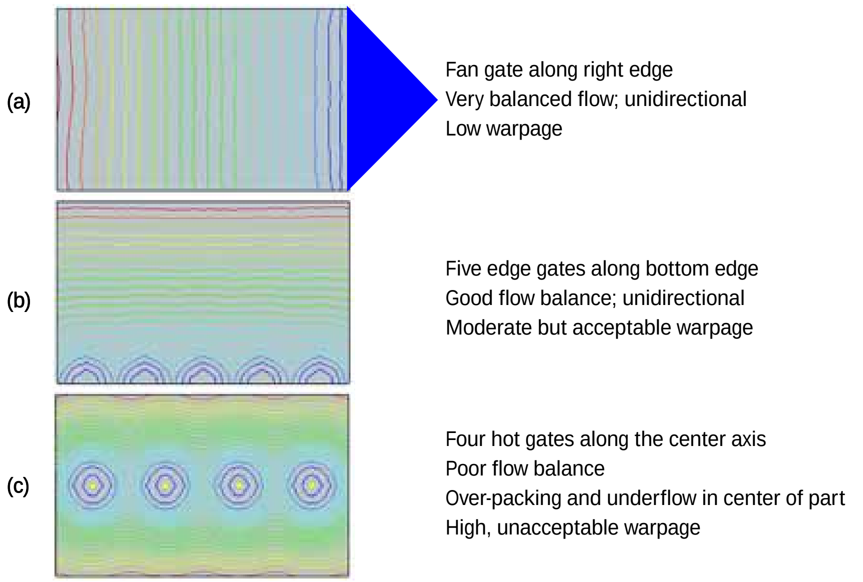


Figure 7.17 Flow patterns with different gate locations

7.3.3 Gate Position

The position or location of the gates is closely tied with the previous two topics. the number of gates, the flow pattern, and position cannot be separated. All are interrelated. There are several considerations for determining a part's gate location, including:

- Place gates to achieve balanced filling—primary importance as discussed above
- Place gates in thicker areas to better pack out the part
- Place gates far from thin features to prevent hesitation
- Place gates against a wall to prevent jetting
- Place gates to prevent weld lines from forming in weak regions of the part or where they will be visible
- Place additional gates to prevent overpacking
- The type of tool being used—is it a two- or three-plate mold?
- Hot or cold runners, or a combination?
- The type of gate desired: edge, tunnel, etc.
- Restrictions on gate location because of part function
- Restrictions on gate location because of tool function

7.3.3.1 Balanced Gate Locations

Throughout this book there has been a discussion of balanced flow within a part. Balanced flow, however, is not necessarily easy to define. Balanced fill is achieved when the extremities of the part fill at the same time and pressure. Below are four examples of gate placement to achieve a balanced flow in a simple rectangle. All have advantages and disadvantages that are discussed.

End-gated Part: The end-gated part, shown in Figure 7.18, is considered balanced because the flow is unidirectional, and material continues to flow through every area of the part once it is filled (with the possible exception of the extreme left corners). Placing a gate on the end of the part will produce an orientation that is aligned down the axis of the part. This type of gate location generally reduces warpage, in particular with amorphous and fiber-filled materials. The disadvantage is that the flow length is quite long, so fill pressures will be relatively high, and packing may be a problem. With constant packing pressures, the variation in volumetric shrinkage can be high but this problem can be overcome with a decaying packing profile.

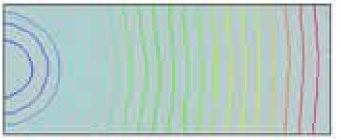


Figure 7.18 End gate with balanced fill

Center-gated Part: The center-gated part in Figure 7.19 is reasonably well-balanced but not as good as the end-gated part. The problem is that the flow front starts out radial, but then straightens out and becomes linear. There is some degree of underflow because the flow length to the middle of the long side is very short compared to the flow length to the long end of the part. This can result in warpage, depending on the material and structure of the part. A center-gate location is usually better for round or square parts.

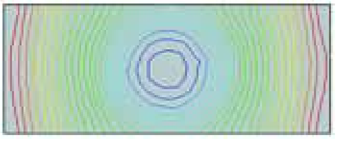


Figure 7.19 Center gate, mostly balanced

Two Gates, Uniform Flow Length: If a second gate is needed, then the positioning of the two gates is critical. In Figure 7.20, the gates are placed so the flow length between the gate and end of the part is the same as the flow length to the weld line. The spacing was calculated by breaking up the length into twice as many sub-moldings as the number of gates. The gates are placed at the boundary between every other sub-molding. This gives the best possible balance within the part with multiple gates. Whenever gates are added to the part, each gate should fill about the same flow length and volume. This is difficult-sometimes impossible-with nonsymmetrical parts, but this should be the initial goal.

One potential problem with this gate location is the weld line. The temperature of the flow front and the pressure on the weld line when it forms determines the quality of the weld line. With this gate location, the weld line forms at the end of fill, therefore the pressure drop between the gate and weld line will be higher than if the gates were closer, and potentially at a lower temperature when the weld line forms.

When considering warpage, this gate configuration will not overpack the center of the part, possibly reducing the warpage of the part.

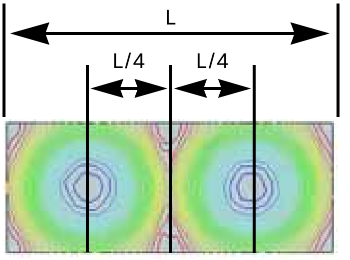


Figure 7.20 Two gates with uniform flow length, mostly balanced

Two Gates, Closer to the Center of the Part: This gate configuration is very similar to the previous one. The gate spacing is calculated by splitting the length into a number of sub-moldings equal to the number of gates plus one. This places the gates closer together and the flow length to the ends of the part is longer, as shown in Figure 7.21.

As a result, there is overpacking between the gates, possibly leading to warpage. The potential for warpage makes this gate configuration less desirable than the one in Figure 7.20; however, the weld line may be of higher quality than the previous case. This is because the weld line is formed closer to the gate at higher temperatures, and will see higher pressures. If quality of the weld line is of primary importance, this gate location may be better than the previous example.

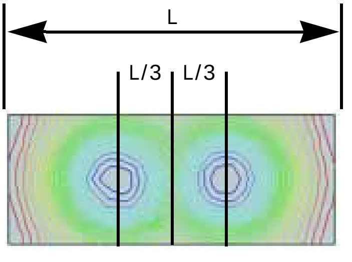


Figure 7.21 Two gates closer to center, not balanced

7.3.3.2 Gate in Thicker Areas

In Figure Figure 7.22 the part has a 5 mm thick section and a 2 mm thick section. An edge gate was placed in the thick section on the part in the center, and in the thin section on the part to the right. Each part has the same gate, runner size, and processing conditions. The results indicate that the part with a gate placed in the thick section has much lower and uniform volumetric shrinkage compared to the part with a gate placed in the thin section.

Depending on the objectives of the part, it may be beneficial to place a gate in a thicker area, even if the balance of the part may not be quite as good. This situation will most likely be the case if the material is semicrystalline and/or if sink marks and voids are critical defects to be avoided.

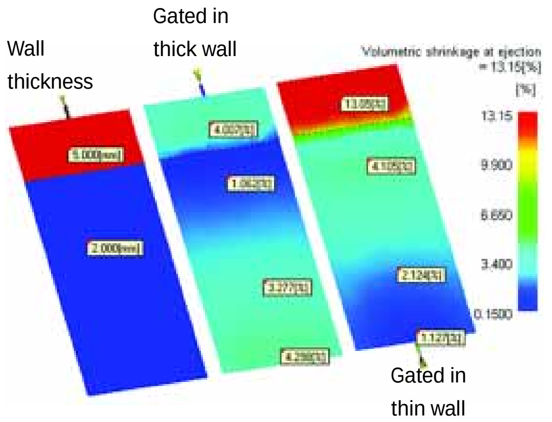


Figure 7.22 Gating in thick and thin sections, comparing volumetric shrinkage

7.3.3.3 Gate Far from Thin Features

When there is a wide variation in wall thickness, place the gate as far as possible from thin features to avoid hesitation. In Figure 7.23 the part has nominal wall is 2 mm and the rib is 1 mm thick. Both examples have the same processing conditions.

In the top part, the gate is close to the thin rib. When the flow front reaches the rib, the flow splits. Polymer flow takes the path of least resistance. Since the pressure required to go into the thick nominal wall is much less than the thin rib, most of the material goes in the nominal wall. The material going into the rib is hesitating so there is little shear heat, and the material gets quite cold and eventually freezes off, creating a short shot.

In the bottom part, the gate is at the far end of the part. In this case, when the material gets to the thin rib, there is not much of the part left to fill. The material hesitates going up the rib, but there is not enough time for the rib to freeze off. The last place to fill is still in the rib, but it does fill. This problem is more likely to occur with semicrystalline materials because they tend to freeze faster.

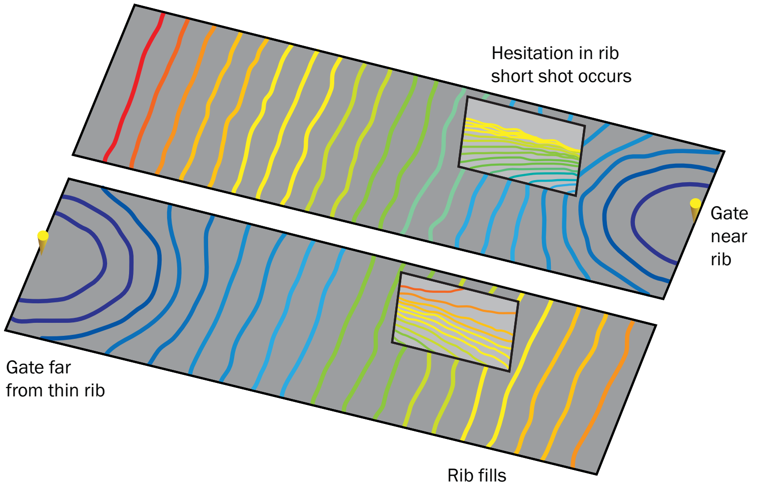


Figure 7.23 Gates near and far from thin feature

7.3.3.4 Add Gates as Necessary to Reduce Pressure

You often need to place gates so that the fill pressure is within the capacity of the injection-molding machine. As a general rule, the maximum fill pressure of the part without a feed system should be about half the machine limit. For a typical machine, the pressure to fill a part should not exceed 70 MPa (10,000 psi).

When a single gate has a pressure that exceeds a pressure limit or guideline, the pressure must be lowered. Changing the gate location to reduce the maximum flow length in the part is a good way to lower the pressure to fill. Once you have achieved the shortest possible or practical flow length and the pressure is still too high, add a second gate.

As you add additional gates, place them so all gates have about the same volume to fill, the same flow length, and a balanced fill. This will reduce the pressure (Figure 7.24).

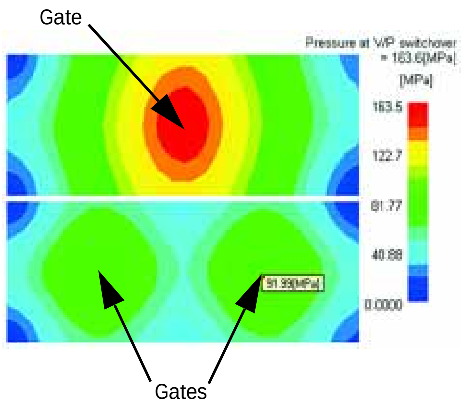


Figure 7.24 Adding a second gate reduces the pressure by reducing the flow length

7.3.3.5 Prevent Overpacking by Adding Gates

Depending on the geometry of the part, adding gates can sometimes improve the packing of the part by improving its uniformity. In Figure Figure 7.25 the single-gated part has a nearly balanced filling pattern. However, due to the center rib close to the gate, the volumetric shrinkage in the rib is very low because it is overpacked. This may cause a problem with warpage, but it also may cause a problem with ejecting the part. There may be other overriding factors in the decision to place a gate, but overpacking may be important. The double-gated part fills the center rib toward the end of fill. The volumetric shrinkage in that center rib is much better than the single gate location.

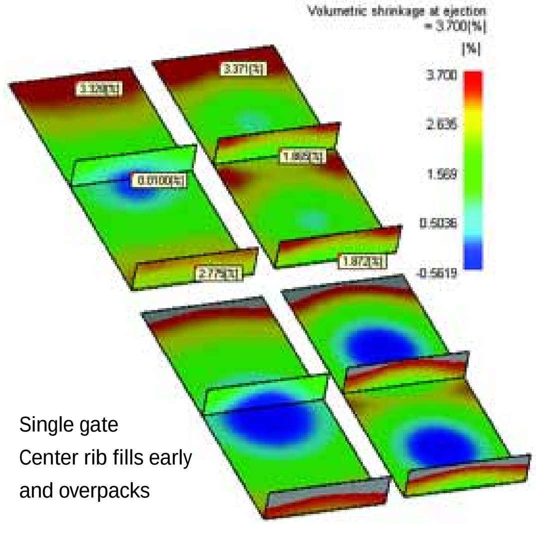


Figure 7.25 Single gate, overpacked rib

7.3.4 Avoiding Common Problems

7.3.4.1 Vent Properly to Prevent Air Traps

To prevent air traps the gate location should allow the air present in the cavity to escape during injection. Failure to vent the air will result in a short shot, a burn mark on the molding, or high filling and packing pressure near the gates.

7.3.4.2 Enlarge the Gate to Avoid Jetting

Gate location and size should prevent jetting (see Section 12.12). Jetting can be prevented by enlarging the gate or by locating the gate in such a way that the flow is directed against a cavity wall.

7.3.4.3 Position Weld and Meld Lines Carefully

The gate location should cause weld and meld lines, if any, to form at appropriate positions that are not objectionable to the function or appearance of the part.

7.3.5 Gate Length

The gate land length for edge-type gates should be as short as possible to reduce an excessive pressure drop across the gate. A suitable gate length ranges from 0.5 to 1.5 mm (0.02 to 0.06 in).

7.3.6 Gate Thickness

Traditionally, gate thicknesses have been determined by general guideline. For most applications and materials the gate thickness is normally 50 to 80% of the gated wall section thickness. For manually trimmed gates, the gate thickness can occasionally be the same as the gated wall section thickness. For automatically trimmed gates, the gate thickness is typically less than 80% of the gated wall section thickness to avoid part distortion during gate breaking. Typical diameters at the gate/part interface for submarine gates range from 0.5 to 2.5 mm (0.02 to 0.10 in). With Moldflow, gates can be more precisely sized using shear rate. Every material in the Moldflow material database has a shear rate limit. Gates should be sized so the shear rate limit of the material is not exceeded. Shear rate limits range from 20,000 1/sec. for PVC to 100,000 1/sec. PP. Most materials are in the 40,000 1/sec. to 60,000 1/sec. range. For highly filled materials, the shear rate should be kept lower when possible because the additives are more sensitive to shear than the polymer itself. In most cases, 20,000 1/sec is low enough.

7.3.7 Freeze-off Time

The freeze-off time at the gate is the maximum effective cavity packing time. If the gate is too large, freeze off might be in the part rather than in the gate. If the gate freezes after the packing pressure is released, flow could reverse from the part into the runner system. A well-designed gate freeze-off time will also prevent backflow of the injected material.

7.4 Using Moldflow for Gate Design

Moldflow is an effective tool for comparing the implications of various gate designs, including:

- Gate location
- Molding window size at the gate location
- Filling pattern
- Gate size based on shear rate

In this example, Moldflow will be used to evaluate a door panel molded from ABS. Three gate locations are to be considered.

7.4.1 Gate Location

The gate locations being considered are shown in Figure 7.26. The part must use edge gates along the perimeter of the part.

7.4.1.1 One End Gate

A single gate location on the end of the part is the preferred gate location with regard to Moldflow design principles. It has unidirectional flow and minimizes the weld lines.

7.4.1.2 One Bottom Gate

The flow length for the end gate is about 1050 mm so the pressure required to fill may be too high. One gate on the bottom of the part reduces the flow length to about 900 mm. The part has mostly unidirectional flow, but there is a much larger radial component.

7.4.1.3 Five Bottom Gates

With the five bottom gates, the flow length is cut still further to about 800 mm. Another advantage is more unidirectional flow because the flow fronts from each gate meet early in the fill and produce a flat flow front. However, the additional gates produce more weld lines that may be objectionable.

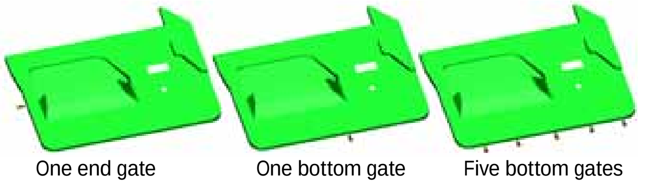


Figure 7.26 Door panel gate locations investigated

7.4.2 Molding Window Size for the Three Gate Locations

As discussed previously, three gating locations were picked based on their advantages relative to Moldflow design principles and their flow length. Because the flow length is very long, the pressure required to fill is a concern. A molding window analysis was run on all three parts to evaluate the processability of the gate locations. The molding window analysis was the first step because running the analysis for all three parts took under 10 minutes.

Figure 7.27 shows the molding windows for the three gate locations. For each gate location, the mold temperature was fixed at 55°C (131°F), the melt temperature ranges from 245 to 265°C (473 to 509°F) and an injection time range of two to 15 seconds. The molding window is split into three possible areas, including:

- Not feasible—requires an injection pressure of more than 80% of the machine capacity. In this case it is 112 MPa (16,240 psi).
- Feasible—requires an injection pressure of more than 50% of the machine capacity or 70MPa (10,150 psi). All other parameters must be within specifications.
- Preferred—requires less than 50% of the machine capacity and all other parameters are within specifications, including shear stress, shear rate, flow front temperature, and clamp force.

Only the five bottom gates option has any preferred molding window at all. This is because of the long flow length in other options, which creates a high injection pressure.

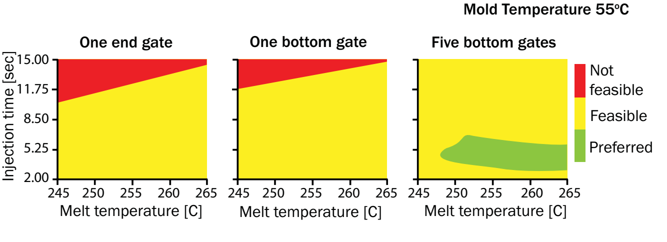


Figure 7.27 Molding windows for the three gate location options

7.4.3 Filling Pattern

From the three original gate locations investigated, only one was found to be possible from a processing standpoint so the filling pattern was investigated on one part only. Figure 7.28 shows the filling pattern for this part. With the five injection locations on the bottom of the part, the flow fronts meet early during the fill and create a flat and unidirectional flow front across the part. The ear on the upper right corner of the part fills last.

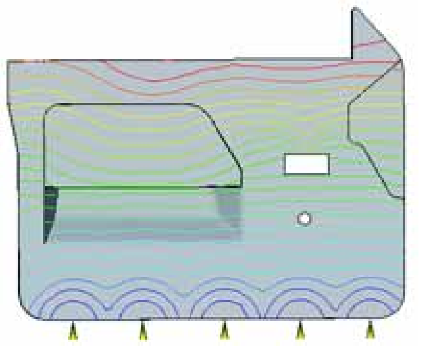


Figure 7.28 Filling pattern on the door panel with five gates on the bottom edge

7.4.4 Gate Size Based on Shear Rate

Once the filling pattern was determined to be acceptable through the part, a feed system was designed to produce the same filling pattern as the fill analysis did without the runners. See Figure 7.29. The original gates have a thickness of 2 mm with the part nominal wall of 3 mm. The original width of the gates was 4 mm.

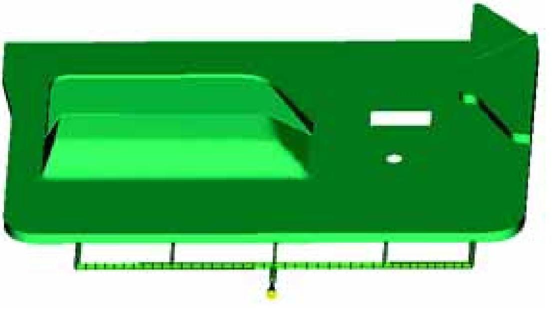


Figure 7.29 Runner system on the door panel with the five gates

The fill analysis with the original gate sizes indicates that the maximum shear rate that is seen in the gates is ~60,000 1/sec. See Figure 7.30. The shear rate limit for the material is 50,000 1/sec. The maximum shear rate is not extremely high, but it should be well below the limit for the material.

Figure 7.30 Shear rate through the gates

The gates were opened to 8 mm wide, cutting the shear rate significantly. Figure 7.31 shows the gate shear rates with the opened gate. The shear rates are now below the limit of the material.

Figure 7.31 Gate shear rates with enlarged gates

8 Runner System Design

- Definitions
- Runner system design principles
- Runner types
- Runner layout
- Initial runner sizing
- Runner balancing
- Using Moldflow for runner balancing

8.1 Definitions

8.1.1 Feed System

Feed system is a generic term used to describe the polymer flow channels in the injection mold from the entrance of the mold to the parts. This term is often used in place of runner system. Feed system is often used because it does not imply hot or cold runner systems—many molds have both.

8.1.2 Runner System

Runner system is a term used to describe the polymer flow channels in the injection mold from the entrance of the mold to the parts. In many cases, this term is used only to describe cold runner systems and not hot runner systems.

In the context for this chapter, the term runner system will be generic and refer to cold runners, hot runners, or a combination of hot and cold runners.

8.1.3 Cold Runner

A *cold runner* is a system of polymer flow channels in an injection mold that are ejected with each cycle of the mold. The mold temperature for cold runners is about the same as the part. See Figure 8.1.

8.1.4 Hot Runner

A *hot runner*, sometimes called a *runnerless* system, is a system of polymer flow channels that are not ejected with each cycle. The hot runners maintain the polymer at a melt temperature approximately equal to the temperature at which it entered from shot to shot. The mold temperature can range from approximately the same mold temperature as the part, to about equal to the melt temperature. See Figure 8.2.

8.1.5 Hot Manifold

A *hot manifold* is a portion of a hot runner system (shown in Figure 8.2) that distributes the polymer from the sprue to the cavities or cold runners, via a hot drop.

8.1.6 Hot Drop

A *hot drop*, sometimes called a *dropper*, delivers the polymer from the hot manifold to the cavities or cold runner. See Figure 8.2.

8.1.7 Sprue

A *sprue* is a portion of the hot or cold runner system. The sprue is the entry point for the polymer into the tool. See Figure 8.1 and Figure 8.2. For cold runner systems, the sprue is tapered, with the smallest end the entry point.

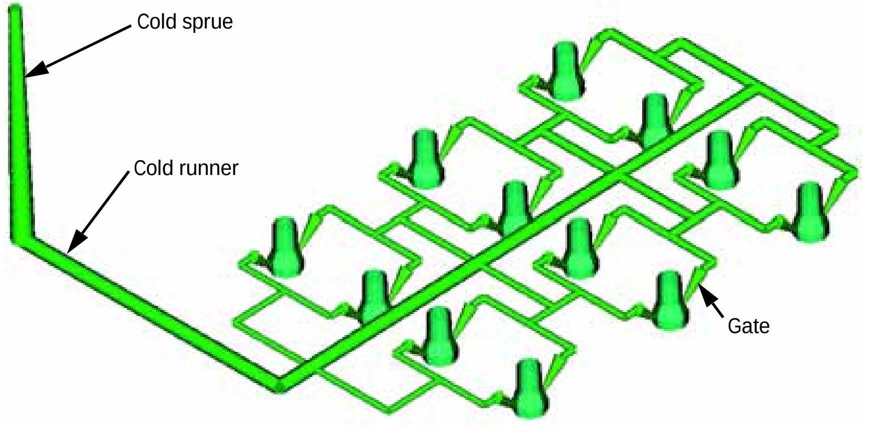


Figure 8.1 Cold runner system

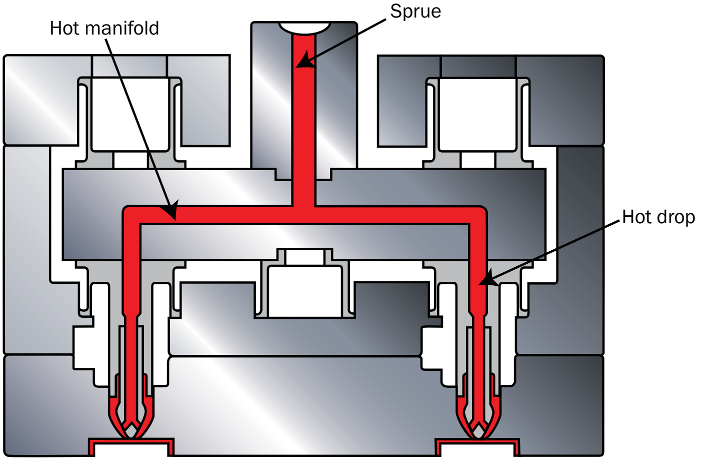


Figure 8.2 Hot runner system

8.2 Runner System Design Principles

8.2.1 Benefits of Good Runner Design

A mold and runner system that has been designed correctly will:

- Have an optimal number of cavities
- Achieve balanced filling of multiple cavities
- Achieve balanced filling of multi-gated cavities
- Minimize scrap
- Eject easily
- Not control the cycle time

8.2.2 Runner Design Philosophy

Traditionally runners have been thought of simply as a means of getting plastic into the cavity. On this basis the size of the runner was not critical, as long as it was big enough to fill the cavity. With the Moldflow philosophy, the design of the runner is crucial because the combination of the position of the gate and the size of the runner controls the filling pattern within the cavity. The runner is used as a flow control device.

8.2.3 Flow Balancing

Runner systems must be designed so each cavity (or portion of a cavity) fills at the same time and pressure. When a runner system is used to balance flow, the total fill pressure of runner plus cavity pressure drop must be equal. It is not sufficient only to balance the runners without considering the cavity. Changing the runner system will alter the cavity pressure drop because the flow rate and frictional heating will change.

The more uniform the balance between cavities and within cavities (multigated part), the higher the part quality will be and the easier the parts will be to mold.

8.2.4 Flow Control

Controlling the flow, or balancing, should be done by runners, not by gates.

8.2.4.1 Gates

Gates are very poor flow control devices because:

- The pressure drop over the gate can be heat-transfer dominated, so any small change in molding conditions gives a large change in filling pattern
- Gates are very prone to hesitation effects
- Entrance and exit losses, which tend to be very unstable, form a high proportion of the total pressure drop
- Machining errors and wear have a major effect on pressure drops

Figure 8.3 shows an example of how gates should not be used to balance the flows paths in a mold. In Figure 8.3 (a), all three parts have the same gate size of 0.50 x 0.50 mm (0.020 x 0.020 in). The gate land is very long. Before the use of flow analysis, balancing of the mold was done by changing the gate size. Since only the left cavity lags significantly behind the other two, the gate size of the left part is opened to 0.75 mm (0.030in) square. The resulting filling pattern is shown in Figure 8.3 (b). Now the left part fills before the other two. There is also a noticeable hesitation between the middle and right parts. Because the middle part is lagging the most, that gate is adjusted next as shown in Figure 8.3 (c). The gate is opened 0.10 mm (0.004 in) in thickness and width. The balance has significantly changed again.

Assuming this process continued until all the parts are filling at about the same time, the molding window would be very small. Any change in the process would have a noticeable influence on the balance between the cavities. This would include natural variation (noise) that occurs on the production floor.

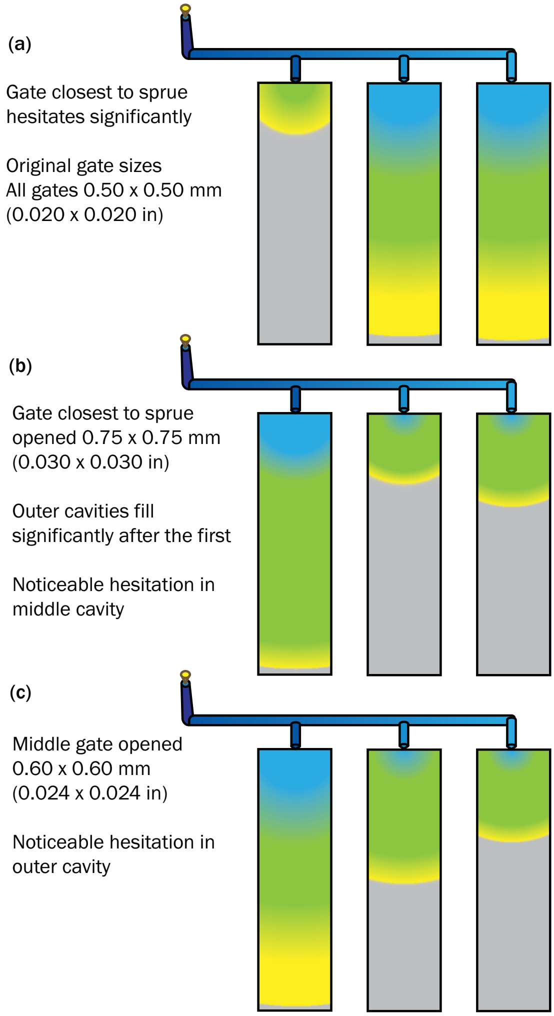


Figure 8.3 Gates make poor flow control devices

8.2.4.2 Runners

Runner systems are very good flow control devices because:

- A runner system is much larger than a gate, and therefore less sensitive to hesitation and thermal effects
- They have a fully developed and stable flow pattern
- Runners are easier to machine accurately

8.2.5 Frictional Heating in Runners

In addition to controlling flow, runners can be used to give controlled frictional heating. The concept of frictional heating in the runners is of major importance. As shown previously in Chapter 2, the residual stress level is lowered proportionately to the raising of the melt temperature. Simply raising the barrel temperature will reduce stress levels, but will also give severe degradation problems. This occurs because the plastic is then subject to a high temperature in the barrel for several machine cycles, a time measured in minutes.

In contrast, running the barrel at a lower temperature and relying on frictional heating in the runner will give the same effect of lower stress levels, but without degradation of the material. This is a result of the plastic only being subject to the higher melt temperature from the time it enters the runner system until it starts to cool, a time measured in seconds.

Using smaller runners generates shear heat, which lowers stress levels in the part and produces a part of higher quality. This allows the barrel melt temperature to be lower while still having the hotter material in the part.

> ✎ Frictional heating is generally associated with cold runners; however, there may be some frictional heating in hot runners as well.

8.2.6 Thermal Shutoff

Runners should be designed to allow for proper filling and packing of the parts without controlling the cycle time. During the compensation phase, molecules are being forced into the cavity as the material is freezing and shrinking. The combination of flowing and freezing at the same time locks in high levels of orientation and residual stress. Larger runners stay open too long allowing more flowing and freezing to occur in parts.

Cold runners should also be small enough so that they do not limit cycle time of the molding machine. The runners do not have to be frozen at ejection, but they must be able to withstand ejection forces. As a maximum, the freeze time of the runner system should be two to three times that of the part.

8.2.7 System and Runner Pressures

In general, the higher the runner pressure drop, the better will be the flow control. The additional frictional heating will lower residual stress levels in the cavity, which should make better parts. The available pressure from the injection machine sets a limit for the maximum total filling pressure. Since normally some safety factor normally is used, the runners are designed so that the total pressure drop, (cavity plus runners), is 70 to 75% of the maximum available injection pressure. However, sink marks always have to be considered.

There is always a conflict between sink marks and stress levels. Stress levels, which effect warping, are minimized by using high runner pressures and high melt temperatures. However, if the runners are made too small, they will freeze off before adequate compensating flow has occurred. Runners, therefore, have to be designed on a cooling time basis to ensure that they freeze off just after the holding pressure is dropped.

8.2.8 Constant Pressure Gradient

Runners should be designed using the constant pressure gradient principle. This will produce the lowest possible volume for a given pressure drop. Runners sized using a constant pressure gradient will get smaller as the runner splits. Figure 8.4 shows an example of a runner system balanced with constant pressure gradient. The model shown is one quarter of the mold. Each time the runner splits, the diameter changes a little.

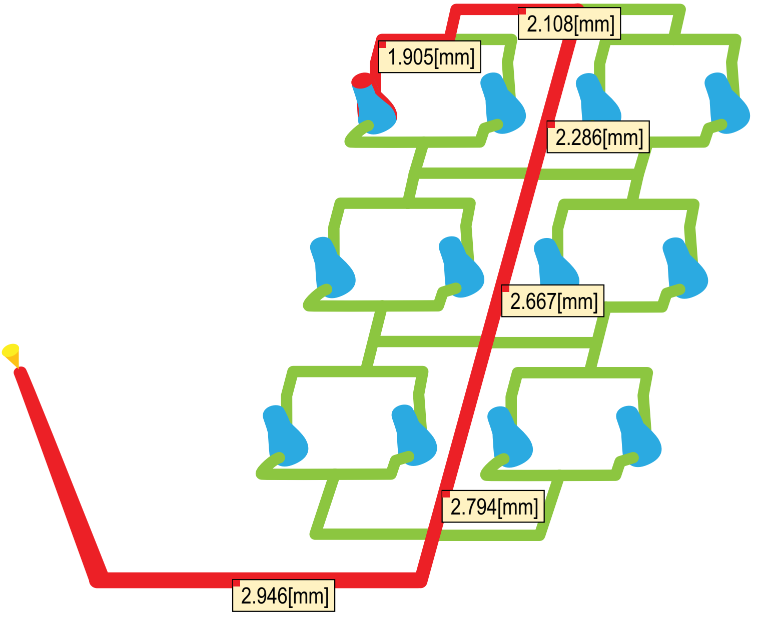


Figure 8.4 Constant pressure gradient used to size and balance the runners

8.2.9 Cold Slug Wells

A cold slug well is an extension of a runner system past the last branch. The purpose of a slug well is to capture the cold slug of polymer that may form in the nozzle between shots. Most of the time, if the slug exists is trapped at the bottom of the sprue. Just in case it is not, the runners are extended slightly. The amount they should extend past the branch is 1.0 to 1.5 diameters, as shown in Figure 8.5.

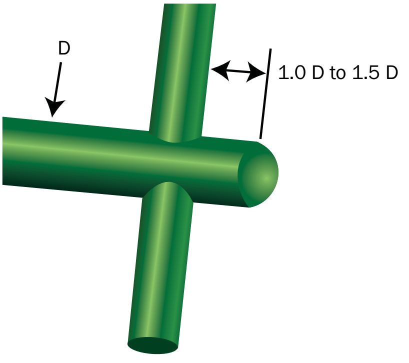


Figure 8.5 Cold slug well

8.2.10 Easy Ejection

Runner design must provide for easy ejection and easy removal from the molded part with proper cross-sectional and draft angle. For most materials, the runner surface must be polished to facilitate flow and part ejection. Extended runner systems should have multiple sprue pullers and ejection locations.

8.3 Runner Types

8.3.1 Cold Runners

Cold runners are very commonly used for injection molds, particularly in smaller molds. They are inexpensive to cut and offer significant flexibility in the design. In the past, not much concern was given to the design of the runners. Now with flow analysis, care should be given to the design of the runners so they produce high quality parts at the lowest possible cycle time. When using automatic runner balancing the sizes produced will not be a standard size. Many times it is possible to change the runners to the nearest standard size. Whe ever runner dimensions are changed a flow analysis should be run to ensure the filling is still balanced.

8.3.2 Hot Runner Systems

Hot runners deliver molten material directly to the part, thus eliminating the cold runners and saving shot volume. In addition to material savings, the cycle time is often faster than cold runners because the cycle time is based on the part rather than the large diameter cold runner. The information in Section 8.2 is primarily based on cold runners, but applies to hot runners as well. When designing hot runner systems, consult with your hot runner vendor to ensure your design will work with the components of the hot runner system. In many cases, standard sizes can be used, but sometimes the required balance demands a nonstandard hot runner component.

There are two types of hot runner systems: insulated and heated.

8.3.2.1 Insulated Runners

Insulated runner molds have oversized passages formed in the mold plate. The passages are of sufficient size that, under conditions of operation, the insulated effect of the plastic (frozen on the runner wall) combined with the heat applied with each shot maintains an open, molten flow path. This type of runner system is not common today because it is difficult to maintain a consistent cycle time.

8.3.2.2 Heated Runners

For heated runner systems, there are two designs: internally heated and externally heated. The first is characterized by internally heated, annulus flow passages, with the heat being furnished by a probe and torpedo located in the passages. This system takes advantage of the insulating effect of the plastic melt to reduce heat transfer (loss) to the rest of the mold. Externally heated runners use a cartridge-heated manifold with circular interior flow passages. The manifold is designed with various insulating features to separate it from the rest of the mold, thus reducing heat transfer (loss). Table 8.8 lists advantages and disadvantages of the three hot runner systems, which are sketched in Figure 8.6.

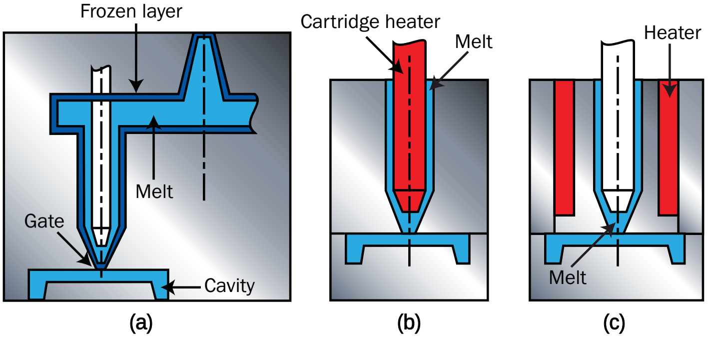


Figure 8.6 Hot runner system types: (a) insulated hot runner, (b) internally heated hot-runner system, and (c) externally heated hot-runner system

Table 8.8: Advantages and disadvantages of hot-runner systems

Hot Runner Type	Advantages	Disadvantages
Insulated	Less complicated design Less costly to build	Undesired freeze-up at the gate Long start-up periods to stabilize melt temperature Problems in uniform mold filling.
Internally Heated	Improved distribution of heat	Moderate cost and complicated design Requires careful balancing and sophisticated heat control Should take into account thermal expansion of various mold components
Externally Heated	Improved distribution of heat Better temperature control	Higher cost and complicated design Should take into account thermal expansion of various mold components

8.4 Runner Layout

8.4.1 Determining the Number of Cavities

8.4.1.1 Factors Involved

The number of cavities depends on the available production time, product quantity required, machine shot size, plasticizing capacities and clamp tonnage capacity, shape and size of the moldings, and mold costs.

8.4.1.2 Formulas

Following are simple formulas for determining the number of cavities. Use the minimum value derived from the following formulas.

Product Quantity: If the dimensional tolerance of the part is not very critical and a large number of moldings are required, multicavity molds are preferred. The number of cavities depends on:

- The time available to supply a specific lot of parts, (*tm*)
- The number of parts in the lot, *L*
- The cycle time to produce a single set of parts, (*tc*).
- The reject factor *K*, expressed as $K = 1/(1 - \text{reject rate})$.

The relation is:

$$\text{Number of cavities} = L \times K \times (tc)/(tm) \tag{8.1}$$

Shot Capacity: The injection machine shot capacity is also a factor in determining the number of cavities by the following:

- 80% of the machine capacity as the shot weight *(S)*
- The part weight *(W)*

The relation is:

$$\text{Number of cavities} = S/W \tag{8.2}$$

Plasticizing Capacity: The injection machine plasticizing capacity is also a factor by the following:

- The plasticizing capacity of the machine *(P)*
- The estimated number of shots per minute *(X)*
- The part weight *(W)*

The relation is:

$$\text{Number of cavities} = P/(X \times W) \tag{8.3}$$

Clamp Tonnage Capacity: The clamp tonnage requirement for a mold is based on:

- Pressure, *(P)*
- Projected area. *(A)*

The relation is:

$$\text{Clamp force} = P \times A \tag{8.4}$$

The pressure needed for an accurate calculation of clamp force is the pressure distribution at its highest value during the filling or packing stages. The clamp force requirements for a part are calculated automatically for a flow analysis in Moldflow. A rough calculation for clamp force would be to take the predicted value of clamp force for one cavity and multiply that by the number of cavities desired and compare that to the clamp force limit of the machine. The best method for determining the clamp force is to use a flow analysis with all the cavities and runner system modeled. Care must be taken when analyzing clamp force. The maximum value of clamp force can change radically depending when the velocity/pressure switchover is done and the injection and packing profile that is used.

8.4.2 Planning the Runner System Layout

8.4.2.1 Basic Layouts

There are three basic runner system layouts typically used for a multicavity system. These layouts are illustrated in Figure 8.7 below.

Standard Runner System: This layout goes by several names, including non-geometrically balanced, herringbone, fish bone, ladder, tree, or artificially balanced. To be artificially balanced, a runner balance analysis must be done to change the size of the secondary runners so all the cavities fill at the same time.

Geometrically Balanced Runner System: This layout is also called naturally balanced or H pattern.

Radial Runner System: This layout is also called a star layout.

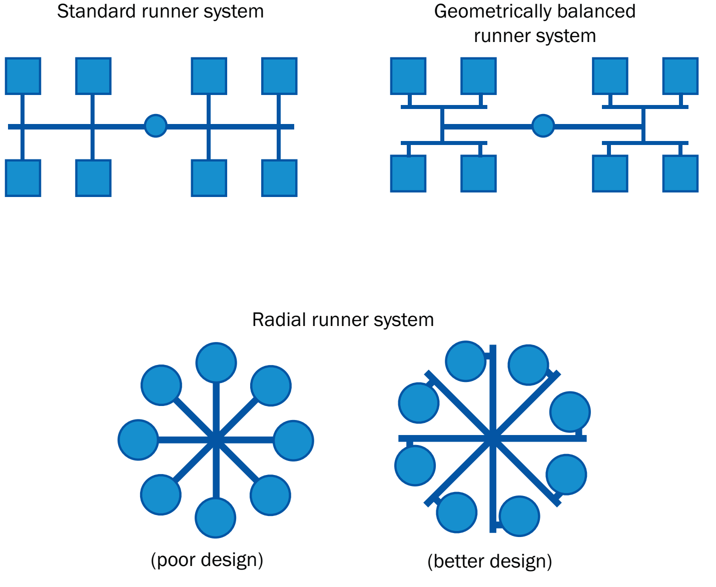


Figure 8.7 Basic runner system layouts

8.4.2.2 Balanced vs. Unbalanced Layouts

Balanced Layouts: The geometrically balanced and radial systems are considered to be balanced. Balanced runners have an equal flow length and runner size from the sprue to all the cavities, so that each cavity fills under the nearly the same conditions. Balanced systems are least sensitive to changes in processing conditions, i.e., there is a large molding window.

Unbalanced Layouts: Although the standard runner system is unbalanced, it can accommodate more cavities than its naturally balanced counterparts with minimum runner volume and less tooling cost. An unbalanced runner system can be artificially balanced by changing the diameter of the runner. Without artificial balancing, the molding window will be very small. Minor changes in processing conditions, in particular fill time will have a huge influence on the balance between cavities. When the runners are balanced, the molding window becomes larger and the mold is easier to run in production, but the molding window will not be as big as a naturally balanced layout.

Automatic Balancing: Runner balancing can be accomplished automatically with Moldflow runner balancing analysis.

8.4.2.3 Artificially Balanced Runner Systems

An artificially balanced runner system will work well if the runner volume is small in relation to the cavity volume, and the variation in the runner sizes is not too large. The balance is maintained by adjusting the pressure drop of a long large-diameter runner against a short small-diameter runner. The pressure drop over the small diameter runner will be much more affected by heat loss than the large diameter runner. Any change in molding conditions will therefore have differing effects on the large and small runners. For example, if the injection rate is reduced, the small runner will be much more affected by heat loss than the large diameter runner. Consequently the cavities on the smaller runner will fill later because the balance has been upset. An artificially balanced runner will therefore only work over a set range of molding conditions. The breadth of this range of molding conditions determines the stability of the molding. Mold stability is an important concept. It indicates whether good parts will be produced, even if molding conditions should vary slightly in production.

Figure 8.8 shows an example of a runner system that cannot be completely balanced because of the cavity and runner layout. The difference in length between the shortest and longest flow part is too great. The length of the runner S_p in the shortest flow path is too short to account for the portion of the longest flow path L_p. The closer the ratio of runner lengths is to 1:1, the easier the runner system will be to balance and the larger the processing window will be.

8.4.3 Partially Balanced Runners

If the runner system does not have sufficient stability with a standard runner system, it may be necessary to use a partially balanced runner system as shown in the drawings in Figure 8.9.

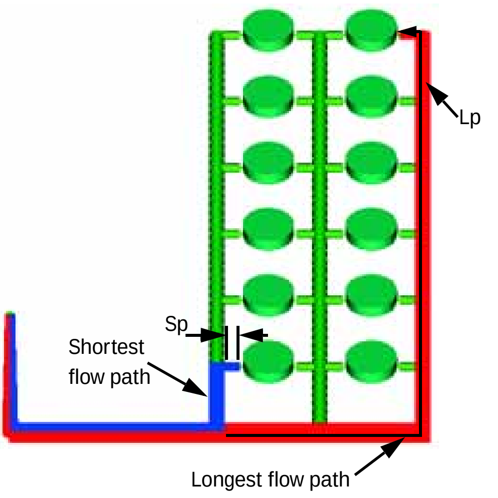


Figure 8.8 Runner system that cannot be balanced

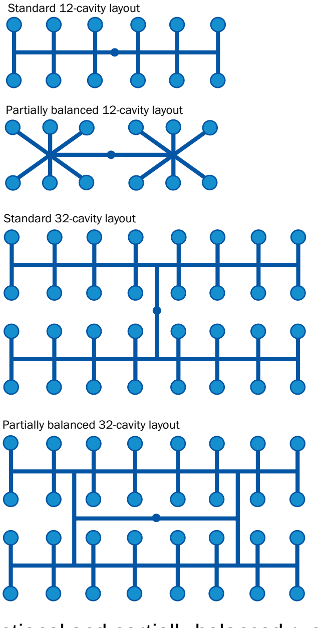


Figure 8.9 Examples of conventional and partially balanced runner layouts

8.4.4 Geometrically Balanced Runners

Today, most single-gated parts in multicavity tools use a geometrically balanced runner layout. This type of runner system has the largest processing window compared to runner layouts that are not geometrically balanced. Traditionally, geometrically balanced runner systems were not as popular because of the volume of the runners and the additional space in the tool necessary. By using Moldflow, the size of the runners can be minimized, allowing the smallest volume of runner in a geometrically balanced configuration.

8.5 Initial Runner Sizing

8.5.1 Determining Sprue Dimensions

Sprues or sprue bushings, are typically standard off the shelf items. For a flow analysis, there are typically three required dimensions:

- The orifice diameter, *O*
- The length, *L*
- The included angle

See Figure 8.10. The orifice diameter is determined by the injection-molding machine's nozzle orifice diameter. The sprue orifice diameter must be slightly larger than the nozzle's diameter so that there is no sharp corner for the polymer to flow over creating an excessive amount of shear. Typically, the sprue orifice is 0.5 mm or 1/32 in (0.031 in) larger than the sprue.

Table 9: Typical standard sprue orifice sizes

Metric Sizes	English Sizes
2.5 mm	3/32 in. (0.094 in.)
3.0 mm	5/32 in. (0.156 in.)
2.5 mm	7/32 in. (0.219 in.)
3.0 mm	9/32 in. (0.281 in.)
4.0 mm	11/32 in. (0.344 in.)
4.5 mm	
6.5 mm	

The length of the sprue is the flow length which is measured from the bottom of the spherical radius to the bottom of the sprue.

The included angle ranges from 1 to 3º, with most sprues with English dimensions having an included angle of 1/2" per foot or 2.38º included angle.

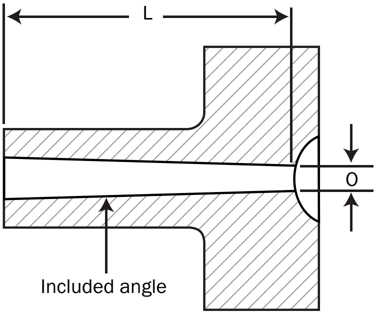


Figure 8.10 Typical sprue dimensions

8.5.1.1 Which Size Sprue to Use

Typically the smallest possible sprue should be used. The smallest diameter is determined by the pressure requirements for the entire tool and the bottom diameter of the sprue compared to the main runner. The pressure drop of the entire tool should be no more than about 75% of the machine capacity. In some cases, sprues can have a pressure drop through them equal to the rest of the tool. In rare cases, the shear rate in the sprue is higher than the gates. This could happen when the tool has many cavities, for example 32. The gate diameter may be small, and the sprue will have a flow rate 32 times that of the gates, assuming one gate per part. The bottom sprue diameter should not be smaller than the diameter of the primary runner it feeds.

8.5.1.2 Nonstandard Sprue Sizes

In some cases, the sprue size is not standard. In these cases, the size of the sprue is determined by the orifice diameter in the same way that standard sprue sizes are, and the diameter of the primary runner the sprue feeds. This is done in cases when using a standard sprue, the bottom diameter of the sprue is much larger than the diameter of the primary runner. When this happens, the sprue normally becomes the limiting factor in determining the cycle time. If a custom sprue is going to be used, the included angle must be great enough to allow for easy ejection of the sprue. Although using a custom sprue may be beneficial, it is rarely done in practice.

8.5.2 Designing Runner Cross Sections

8.5.2.1 Common Designs

There are several common runner cross-sectional designs. They are illustrated in Figure 8.11.

- Full-round runner
- Trapezoidal runner
- Modified trapezoidal runner
- Half-round runner
- Rectangular runner

8.5.2.2 Recommended Cross-sectional Designs

The first three runner cross-sectional designs listed above are generally recommended.

Full-round Runner: The full-round runner is the best in terms of a maximum volume to-surface ratio, which minimizes pressure drop and heat loss. However, the tooling cost is generally higher because both halves of the mold must be machined so that the two semicircular sections are aligned when the mold is closed.

Trapezoidal Runner: The trapezoidal runner also works well and permits the runner to be designed and cut on one side of the mold. It is commonly used in three-plate molds, where the full-round runner may not be released properly, and at the parting line in molds, where the full-round runner interferes with mold sliding action. The shape of the trapezoid is critical. Figure 8.12 shows the proper shape of a trapezoidal runner compared to a round cross section. The depth of the trapezoid is equal to the diameter of the runner, and the angled sides are tangent to a circle. The included angle is normally between 10 to 20º, or the taper angle is half the included angle.

Modified Trapezoidal Runner: This cross section is a combination of round and trapezoidal shapes. The bottom of the runner is fully round and extends to the parting line at the included angle of the trapezoid.

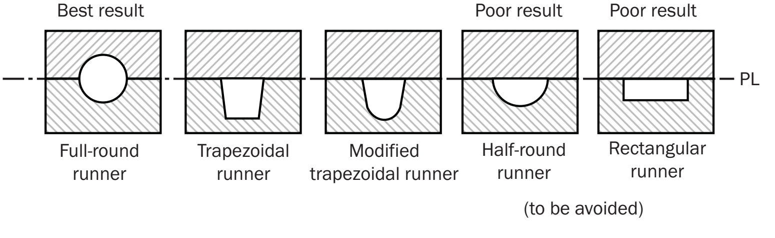


Figure 8.11 Commonly used runner cross sections

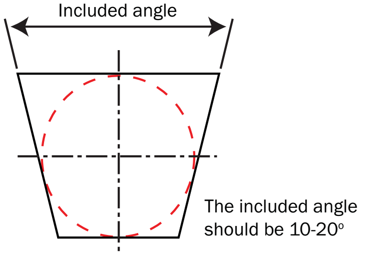


Figure 8.12 Trapezoidal runner shape

8.5.2.3 Hydraulic Diameter and Flow Resistance

To compare runners of different shapes, use the hydraulic diameter, which is an index of flow resistance. The higher the hydraulic diameter, the lower the flow resistance. Hydraulic diameter can be defined as:

$$D_h = \frac{4A}{P} \tag{8.5}$$

where

D_h = hydraulic diameter

A = cross-sectional area

P = perimeter

Figure 8.13 illustrates how to use the hydraulic diameter to compare different runner shapes.

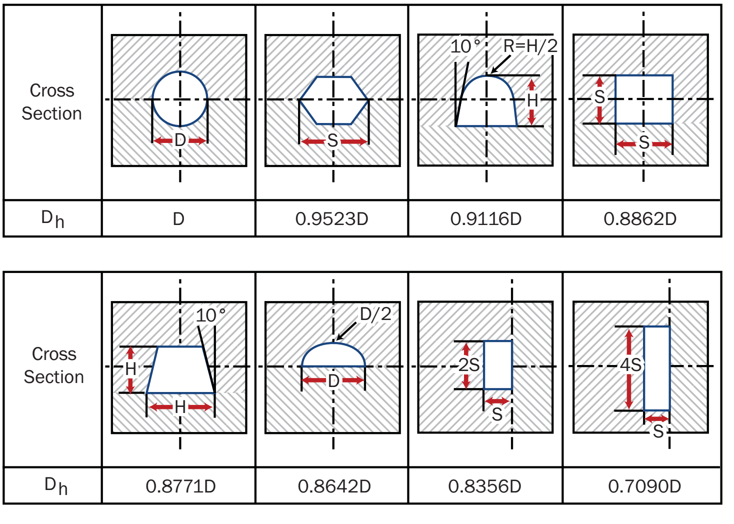


Figure 8.13 Equivalent hydraulic diameters

8.5.3 Determining Runner Diameters

A flow analysis is the best place to determine or optimize the runner diameter. However, where is a good starting point? The pressure drop over the runner is related to:

- Viscosity of the material
- Flow length in the runner
- Volumetric flow rate of the polymer

As any of the items listed above increase, so does the pressure requirement.

8.5.3.1 Typical Runner Diameters

Over the years, guidelines for runner sizes have been developed by many different organizations. Generally, they are all about the same. Most give a wide range of sizes for a given material type. This can be used as a good starting point. Table 10 lists typical runner diameters for unfilled materials.

8.5.3.2 Branched Runners

In geometrically balanced runner systems, it is common for the runners to reduce in size from the sprue to the gates. The change in size would occur when the runners split or branch. Figure 8.14 shows an example of changing the runner size. It is best to calculate the runner sizes using the constant pressure gradient principle using Moldflow runner balancing analysis. However, the sizes can be approximated using the following formula:

$$d_{feed} = d_{branch} \times N^{1/3} \tag{8.6}$$

where:

d_{feed} = the diameter of the runner feeding the branch.

d_{branch} = the diameter of the runner branch.

N = the number of branches.

In a geometrically balanced runner system, the number of branches will always be two.

For the model in Figure 8.14, the diameter of the runner at the gate is 3.0 mm and is the starting point for the calculations. The number of cavities is eight, so there are two branches in the runner system. To calculate the secondary runner:

$$3.78\text{mm} = 3\text{mm} \times 2^{1/3} \tag{8.7}$$

The primary runner is calculated based on the secondary runners and is 4.76 mm.

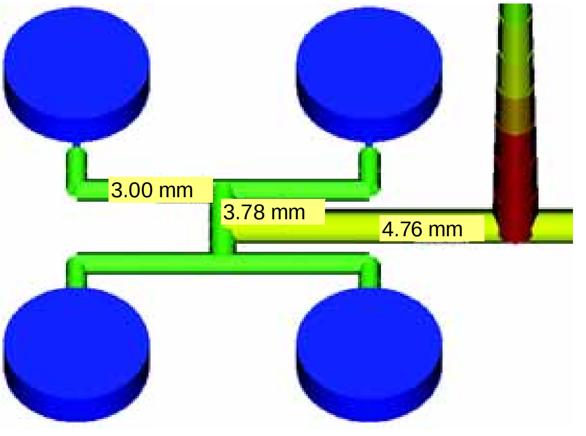


Figure 8.14 Runner diameters calculated based on branching

Table 10: Typical runner diameters for unfilled generic materials

Material	Diameter		Material	Diameter	
	mm	inch		mm	inch
ABS, SAN	5.0-10.0 3	3/16-3/8	Polycarbonate	5.0-10.0	3/16-3/8
Acetal	3.0-10.0	1/8-3/8	Thermoplastic polyester (unreinforced	3.0-8.0	1/8-5/16
Acetate	5.0-11.0	3/16-7/16	Thermoplastic polyester (reinforced)	5.0-10.0	3/16-3/8
Acrylic	8.0-10.0	5/16-3/8	Polyethylene	2.0-10.0	1/16-3/8
Butyrate	8.0-10.0	5/16-3/8	Polyethylene	2.0-10.0	1/16-3/8
Fluorocarbon	5.0-10.0	3/16-3/8	Polyphenylene oxide	6.0-10.0	1/4-3/8
Impact acrylic	8.0-13.0	5/16-1/2	Polypropylene	5.0-10.0	3/16-3/8
Nylon	2.0-10.0	1/16-3/8	Polysulfone	6.0-10.0	1/4-3/8
Phenylene	6.0-10.0	1/4-3/8	Polyvinyl (plasticized)	3.0-10.0	1/8-3/8
Phenylene sulfide	6.0-13.0	1/4-1/2	PVC Rigid	6.0-16.0	1/4-5/8
Polyallomer	5.0-10.0	/16-3/8	Polyurethane	6.0-8.0	1/4-5/16

8.6 Runner Balancing

8.6.1 How Runner Balancing Works

To conduct a runner balance analysis, the filling of the part must be optimized including molding conditions:

- Mold temperature
- Melt temperature
- Injection time

A filling analysis is run with the preliminary runner layout and diameters. This analysis checks how far out of balance the starting point is and provides information on the target balance pressure. Runner balancing is driven by balance pressure. The higher the balance pressure, the smaller the runner diameters.

8.6.2 When Are the Runner Sizes Optimized?

When doing a runner balance, there are generally two primary goals:

- Ensure that all the cavities fill at the same time (balanced)
- The runner volume is made small as possible

The second goal is typically more work than the first.

> ✎ For geometrically balanced runner systems, the balancing process just optimizes the runner volume.

8.6.3 Validating the Balance

There are several levels of validation that can be done to ensure the balance is good.

8.6.3.1 Pressure

The pressure to fill the mold should be no more than about 75% of the machine's injection pressure capacity. If the pressure is well above this amount, the parts may be difficult to mold consistently at high quality.

8.6.3.2 Temperature

The temperature entering the parts should be close (within 1 to 3ºC) to the temperature that was defined as optimum for the part. Because runners will have shear heat, the temperature entering the sprue should be lowered so that by the time the polymer reaches the cavity, it is at the optimized temperature.

8.6.3.3 Freeze Time

The freeze time of the runners should not be too small or too large. Normally the smallest runner section dominates. The shortest cooling time of a runner should be no less than 80% of the freeze time for the part near the gate. If the part has a high tolerance or tight tolerances for sink marks, the minimum cooling time should be about 100% of the part’s. The largest runner should have a cooling time of no more than two to three times that of the part. Meeting both guidelines is normally very difficult. The small runner is most critical because it relates to part quality.

8.6.3.4 Volumetric Shrinkage

The above methods of validation are done with a filling analysis. The next level is to run a packing analysis. The volumetric shrinkage should be nearly uniform between all the cavities. This becomes most critical for family tools. With most family tools, the parts assemble together. If the volumetric shrinkage is not the same between the parts, they will not assemble well. Some tools may need to have an unbalanced fill so the volumetric shrinkage can be made uniform. This problem would indicate that the molding window for the mold is rather small.

8.6.3.5 Linear Shrinkage and Warpage

The final step of verification is running a warpage analysis. From this analysis, linear dimensions can be checked to ensure each cavity is within tolerance. Deflections can be checked to ensure that all the parts warp satisfactorily.

8.6.4 Processing Window

The size of the processing window should be investigated. Whenever a tool is artificially balanced, the processing window will get bigger compared to not being balanced, but it will be still be smaller than a geometrically balanced tool. The primary variable that will influence the processing window is injection time or flow rate. Several analyses should be run at injection times both faster and slower than the optimum to determine how much variation is tolerable.

Generally, a faster injection time will allow the parts closer to the sprue to fill first. If the injection time is increased, the cavities further from the sprue will fill first.

8.6.4.1 Material Used

A runner balance is done with a specific polymer. If the runner balance was done with one material, and the parts are molded with another material—even in the same family of materials—the balance will not be the same. Running a fill analysis with different materials with the “balanced” runner system is another way to test the sensitivity of the tool.

8.7 Using Moldflow for Runner Balancing

Below are three examples of using Moldflow to size and balance runner systems. The examples include:

- A 48-cavity tool with each part having two gates
- A two-cavity family tool
- A multigated part with three gates

8.7.1 Runner Balancing a 48-cavity Tool

Figure 8.15 shows an example of one quadrant of a 48-cavity seal tool before and after the runners were redesigned and balanced. The parts have two gates on them to help maintain roundness. The original runner design could not be balanced. The difference in flow length was too great. The cavity layout was changed to have four rows of three parts. Now there are four areas of the runners that are naturally balanced within the area.

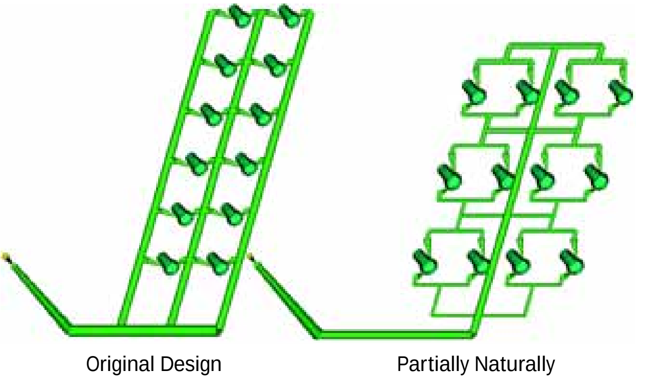


Figure 8.15 Multi-cavity tool runner layouts before and after balancing

Figure 8.16 shows the filling pattern for both designs. The fill time in the original design shows that the cavities close to the sprue fill in about 0.4 seconds when only 65% of the shot volume has filled. The revised design shows that all the parts fill within about 0.01 seconds. Finally, Figure 8.17 shows the difference in volumetric shrinkage. The original design has a shrinkage range of about 2.8% while the revised design with the partially naturally balanced runners has shrinkage range of about 0.25%, comparing the same area on each part. The total range of shrinkage for the original design is 4.1% and for the revised design is 2.0%.

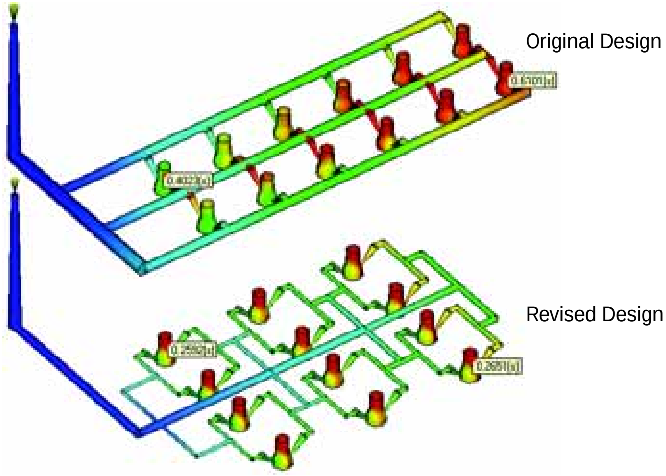


Figure 8.16 Fill time before and after the runner optimization

Figure 8.17 Volumetric shrinkage before and after the runner optimization

8.7.2 Runner Balancing for a Family Mold

To balance the runners for a family tool the parts themselves need to be optimized. This is a particular challenge for family tools because the molding conditions need to be the same. Generally finding an injection time that works well for both parts is difficult. The molding window analysis quickly identifies molding conditions that will work for both parts. A filling analysis on each part will validate the molding conditions.

Figure 8.18 shows the before-and-after filling patterns of a box and lid family tool. Before the runner balance, the box was about half full when the lid fills. After the balance, the box fills first, but only slightly. The runner balance sizes the runner going to the lid that is a bit smaller than the entrance of the drop going to the part. Having a small runner feeding a larger drop is generally not considered a good idea. Alternatives could be:

- Reduce the target pressure used for the runner balance to open up the runners.
  - The problem with this solution is that the runners would get much larger in diameter. As the balance pressure gets lower, the change in diameter becomes significant. The runner volume goes up and so will the cycle time. Neither is a good decision economically.
- Step the parting line so the top of the box and lid are at the same Z location and the drop lengths are the same.
  - This method would probably save material; however, the runner to the lid would have to get much smaller than it is now to maintain the balance. Possibly the tool would be more difficult to balance and more importantly would have a smaller molding window. Of the two alternatives listed, this is more feasible.

The advantage of using flow analysis is that design alternatives can be tested quickly and inexpensively compared to cutting steel and conducting a mold trial.

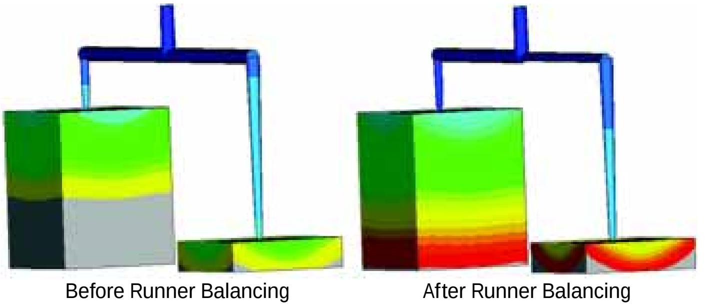


Figure 8.18 Family tool filling time before and after runner balance

8.7.3 Runner Balancing for a Multigated Part

When working with large multigated parts, many things need to be considered. The list changes depending on the part but there are generally several key issues:

- Determine the number of gates
- Achieve a balanced fill
- Achieve a unidirectional fill
- Position weld lines in the least sensitive areas possible

The list could have several more issues on it. In the case of the part shown in Figure 8.19, the flow length was quite large, so both injection pressure and clamp tonnage were problems. Because there needed to be as few weld lines as possible, the number of gate locations were kept to a minimum. Air traps are a real possibility with multigated parts like this. Care was required to ensure air traps were not formed in unvented areas.

Once the gate locations were determined, the runner system needed to be created. In this case, it was a hot runner system. Often, the exact position of the gates is determined by how the hot manifold needs to be created. Once the hot runner system is added to the model, the size of the hot drops needs to be adjusted to ensure the filling pattern in the part is still acceptable.

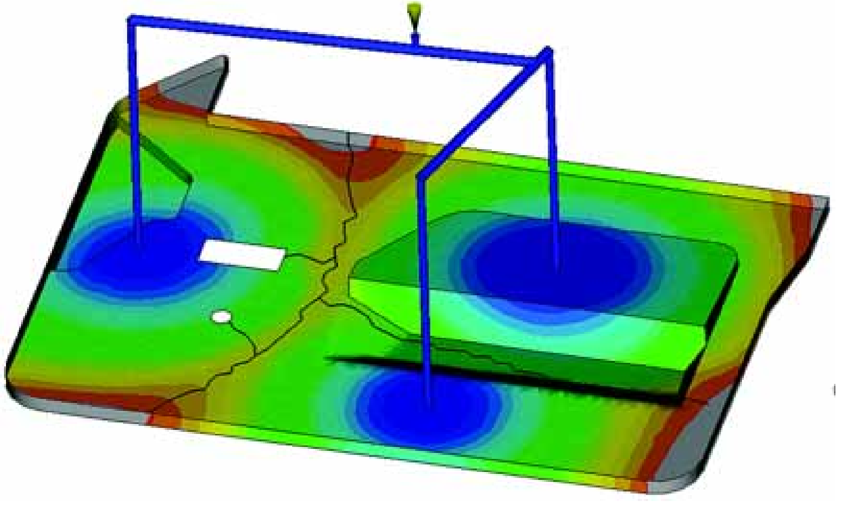


Figure 8.19 Door panel filling pattern and weld/meld lines

9 Cooling System Design

- Mold cooling system overview
- Cooling-channel configuration
- Alternative cooling devices
- Cooling system equations
- Design rules
- Using Moldflow for cooling system design

9.1 Mold Cooling System Overview

9.1.1 Importance of Cooling System Design

Mold cooling can account for more than two-thirds of the total cycle time in the production of injection-molded thermoplastic parts. Figure 9.1 illustrates this point. An efficient cooling circuit design reduces the cooling time, which in turn increases overall productivity. Moreover, uniform cooling improves part quality by reducing residual stresses and maintaining dimensional accuracy and stability (see Figure 9.2).

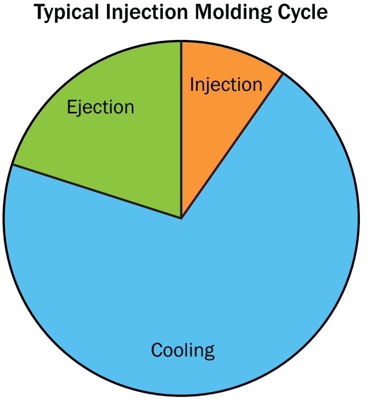


Figure 9.1 Mold cooling accounts for more than two-thirds of the total cycle time

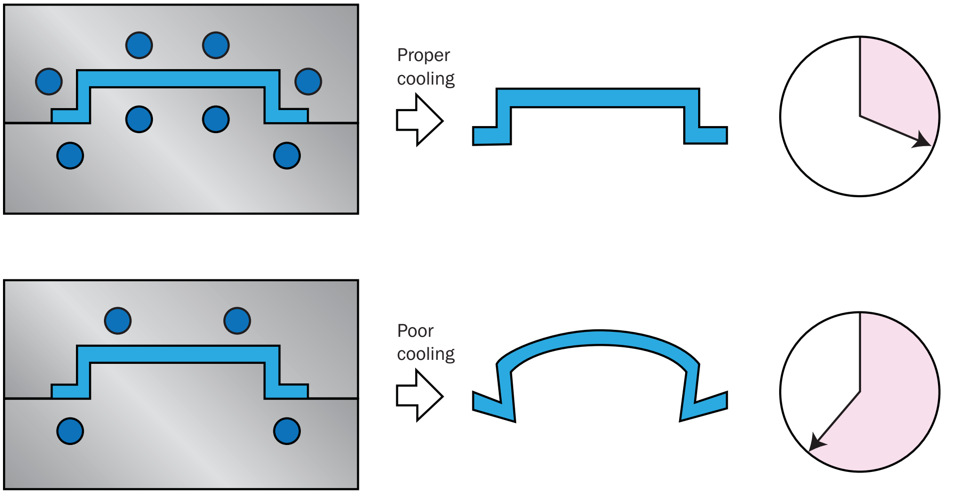


Figure 9.2 Proper and efficient cooling improves part quality and productivity

9.1.2 Mold Cooling System Components

A mold cooling system typically consists of the following items:

- Temperature controlling unit
- Pump
- Supply manifold
- Hoses
- Cooling channels in the mold
- Collection manifold

The mold itself can be considered as a heat exchanger, with heat from the hot polymer melt taken away by the circulating coolant. Figure 9.3 and Figure 9.4 illustrate the components of a typical cooling system.

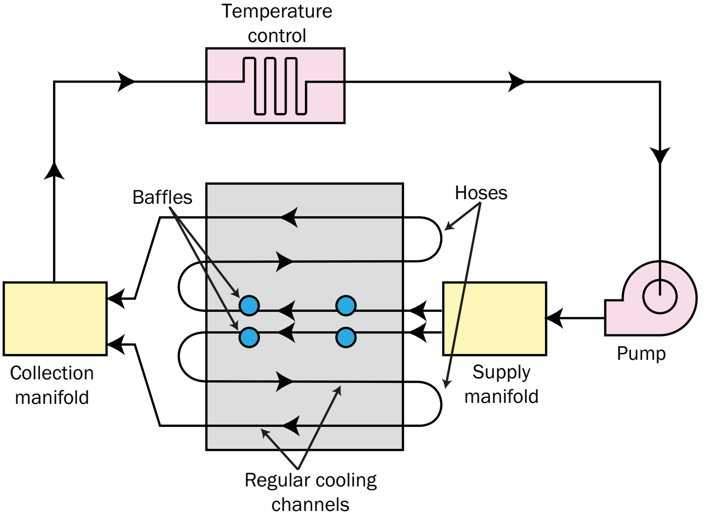


Figure 9.3 A typical cooling system for an injection-molding machine

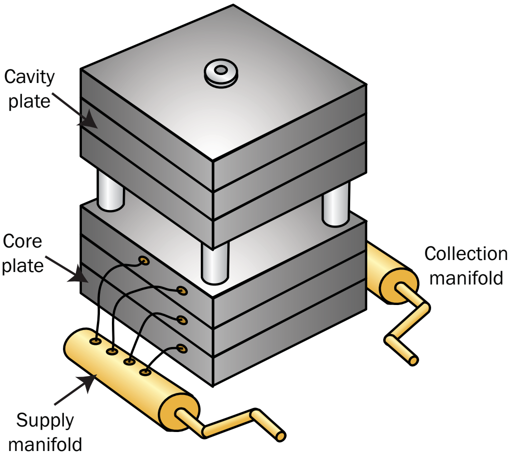


Figure 9.4 A cooling channel assembly attached to the mold plates

9.2 Cooling-channel Configuration

9.2.1 Types of Cooling Channels

Cooling-channel configurations can be series or parallel as illustrated in Figure 9.5 below.

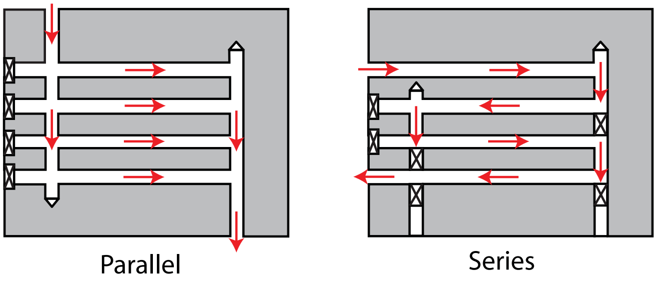


Figure 9.5 Cooling-channel configurations

9.2.1.1 Parallel Cooling Channels

Parallel cooling channels are drilled straight through from a supply manifold to a collection manifold. Due to the parallel design flow characteristics, the flow rate along various cooling channels will be different because of each individual channel's flow resistance differences. These varying flow rates in turn cause the heat transfer efficiency of the cooling channels to vary from one to another. As a result, cooling of the mold will not be uniform with a parallel cooling-channel configuration. Figure 9.6 shows an example of how flow rate and Reynolds number are influenced by parallel cooling channels versus series cooling channels. The example has 10 branches in the parallel system with one to seven baffles in the branch. The inlets and outlets are on the same side of the tool to facilitate quick mold changes. With a flow rate of 25 liters/min. the flow rate in the branches ranges from 1.8 liters/min. to 4.4 liters/ min. The Reynolds number ranges from 4531 to 11280.

If the 10 branches are hooked up in series, and the flow rate is cut to 2.5 liters/min. the Reynolds number in all the branches is 6442. The flow rate was cut from 25 liters/min. to 2.5 liters/min. in the series circuit to give the same flow rate the circuit as would be in all the branches if the flow rate in the parallel circuit would split equally. It is clear that the parallel circuit leads to very poor flow rates and Reynolds numbers leading to nonuniform cooling in the tool. Reynolds numbers are discussed further in Section 9.4.2 and Section 9.5.3.3.

Typically, the cavity and core sides of the mold each have their own system of parallel cooling channels. The number of cooling channels per system varies with the size and complexity of the mold.

9.2.1.2 Series Cooling Channels

Cooling channels connected in a single loop from the coolant inlet to its outlet are called series cooling channels. This type of cooling-channel configuration is the most commonly recommended and used. By design, if the cooling channels are uniform in size, the coolant can maintain its (preferably) turbulent flow rate through its entire length. Turbulent flow enables heat to be transferred more effectively. Heat transfer of coolant flow is discussed more thoroughly in Section 9.5.3. However, you should take care to minimize the temperature rise of the coolant because the coolant will collect all the heat along the entire cooling-channel path.

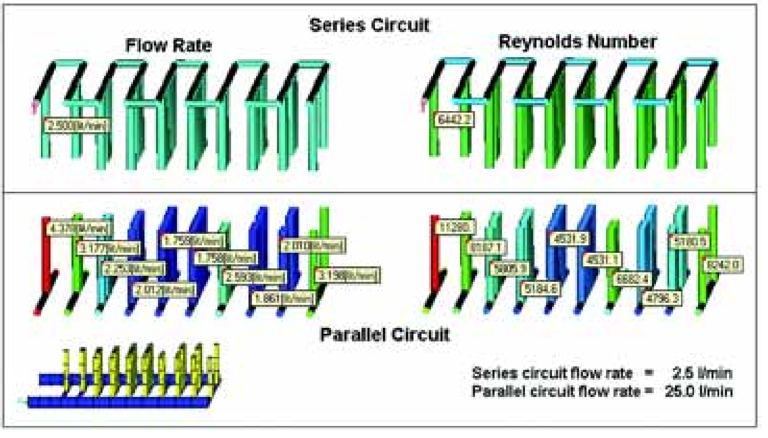


Figure 9.6 Flow rate and Reynolds number differences in series and parallel circuits

9.3 Alternative Cooling Devices

9.3.1 What Do They Do?

Baffles and bubblers are sections of cooling lines that divert the coolant flow into areas that would normally lack cooling. Normal cooling channels are typically drilled straight through the mold's cavity and core. The mold, however, may consist of areas that cannot be addressed by regular cooling channels. Alternate methods for cooling these areas uniformly with the rest of the part involve the use of baffles, bubblers, or thermal pins, as shown in Figure 9.7 below.

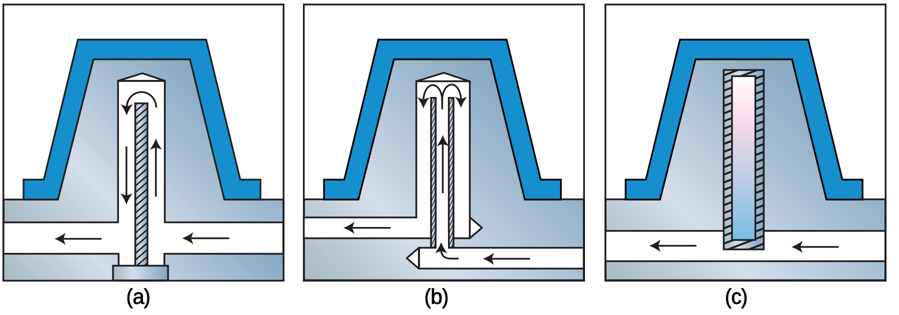


Figure 9.7 (a) Baffle (b) Bubbler (c) Thermal pin

9.3.2 Baffles

A baffle is actually a cooling channel drilled perpendicular to a main cooling line with a blade separating one cooling passage into two semicircular channels. The coolant flows in one side of the blade from the main cooling line, turns around the tip to the other side of the baffle, and then flows back to the main cooling line.

The best baffle designs have the diameter of the baffle larger than the diameter of the channel feeding it. This is done for two reasons: first, to ensure the blade diverting the flow completely blocks the channel so the entire flow goes up the baffle; and, second, so the flow cross section of the baffle is similar to that of the supply channel and not less than half the size. The temperature distribution on one side of the baffle's blade may differ from that on the other side. This can be eliminated if the brass blade (or some other non-ferris metal) forming the baffle is twisted. For example, the helix baffle, as shown in Figure 9.8 (a), conveys the coolant to the tip and back in the form of a helix. It is useful for diameters of 12 to 50 mm, and makes for a very homogeneous temperature distribution. Another logical development of baffles are single- or double-flight spiral cores, as shown in Figure 9.8 (b).

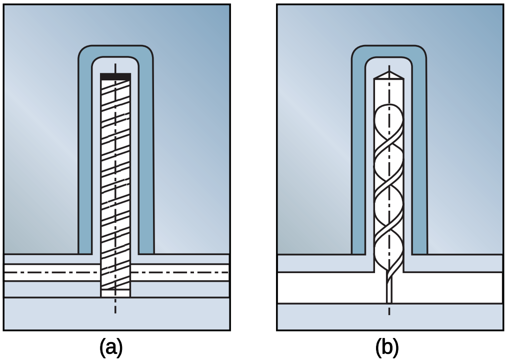


Figure 9.8 (a Helix baffle (b) Spiral baffle

9.3.3 Bubblers

A bubbler is similar to a baffle except that the blade is replaced with a small tube. The coolant flows into the bottom of the tube and “bubbles” out of the top, as does a fountain. The coolant then flows down around the outside of the tube to continue its flow through the cooling channels.

The most effective cooling of slender cores is achieved with bubblers. The diameter of both must be adjusted in such a way that the flow resistance in both cross sections is equal. The condition for this is:

$$\frac{\text{Inner Diameter}}{\text{Outer Diameter}} = 0.707 \tag{9.1}$$

Bubblers are commercially available and are usually screwed into the core, as shown in Figure 9.9. Up to a diameter of 4 mm, the tubing should be beveled at the end to enlarge the cross section of the outlet. Bubblers can be used not only for core cooling, but are also for cooling flat mold sections, which cannot be equipped with drilled or milled channels.

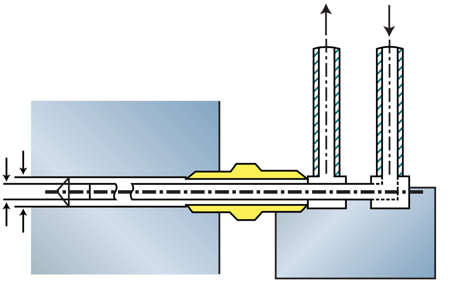


Figure 9.9 Bubblers screwed into core

Because both baffles and bubblers can have narrowed flow areas, the flow resistance increases. Therefore, care should be taken in designing the size of these devices. The flow and heat transfer behavior for both baffles and bubblers can be readily modeled and analyzed by Moldflow cooling analysis.

9.3.4 Thermal Pins

A thermal pin is an alternative to baffles and bubblers. It is a sealed cylinder filled with a fluid. The fluid vaporizes as it draws heat from the tool steel and condenses as it releases the heat to the coolant, as shown in Figure 9.10. The heat transfer efficiency of a thermal pin is almost 10 times as great as a copper tube. For good heat conduction, avoid an air gap between the thermal pin and the mold, or fill it with a highly conductive sealant.

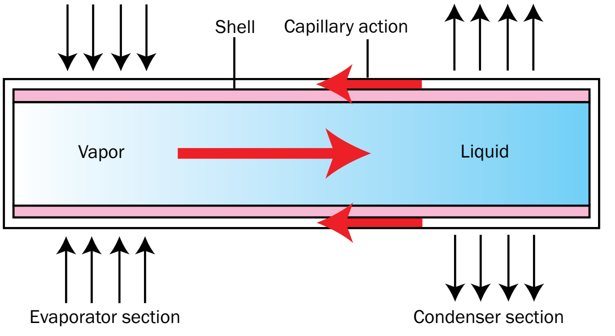


Figure 9.10 Thermal pin heat transfer efficiency

9.3.5 Cooling Slender Cores

If the diameter or width is very small (less than 3 mm), only air cooling is feasible. Air is blown at the cores from the outside during mold opening or flows through a central hole from inside, as shown in Figure 9.10. This procedure, of course, does not permit maintaining an exact mold temperature and is generally not recommended. This method should only be used if no other means of cooling the core can be done.

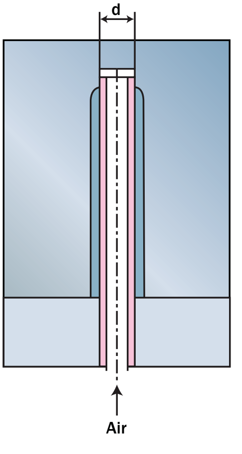


Figure 9.11 Air cooling of a slender core

Better cooling of slender cores (those measuring less than 5 mm) is accomplished by using inserts made of materials with high thermal conductivity, such as copper alloys. This technique is illustrated in Figure 9.12. Such inserts are press-fitted into the core. The inserts should extend into the mold base and should have a cooling channel pass through or touch the insert.

When the high thermal conductivity insert touches the coolant, the efficiency of the heat transfer is very high.

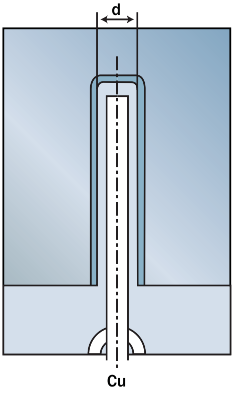


Figure 9.12 Using high thermal conductivity material to cool a slender core

9.3.6 Cooling Large Cores

For large core diameters (40 mm and larger), a positive transport of coolant must be ensured. This can be done with inserts in which the coolant reaches the tip of the core through a central bore and is led through a spiral to its circumference and between core and insert helically to the outlet, as shown in Figure 9.13. This design weakens the core significantly.

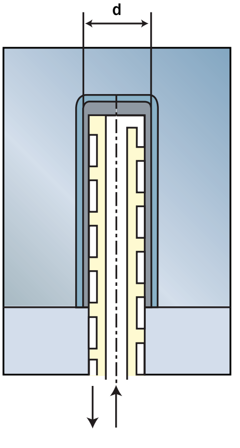


Figure 9.13 Use of helical baffle to cool large core

9.3.7 Cooling Cylinder Cores

Cooling of cylinder cores and other round parts should be done with a double helix, as shown in Figure 9.14. The coolant flows to the core tip in one helix and returns in another helix. For design reasons, the wall thickness of the core should be at least 3 mm in this case.

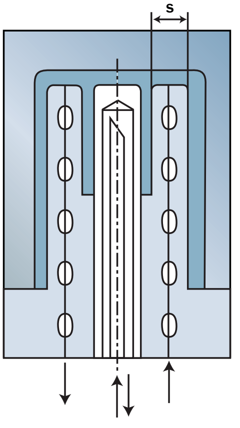


Figure 9.14 Double helix with center bubbler

9.4 Cooling System Equations

9.4.1 Cooling Time

Theoretically, cooling time is proportional to the square of the heaviest part wall thickness or the power of 1.6 for the largest runner diameter. That is:

$$\text{Cooling Time} \propto \frac{(\text{heaviesst wall thickness})^2}{(\text{thermal diffusivity of polymer melt})} \tag{9.2}$$

$$\text{Cooling Time} \propto \frac{(\text{Largest runner diameter})^{1.6}}{(\text{thermal diffusivity of polymer melt})} \tag{9.3}$$

where the thermal diffusivity of polymer melt is defined as:

$$\text{thermal diffusivity} = \frac{(\text{Thermal conductivity})}{(\text{density})\ (\text{specfic heat})} \tag{9.4}$$

In other words, doubling the wall thickness quadruples the cooling time.

9.4.2 Reynolds Number and Coolant Flow

Whether or not the coolant flow is turbulent can be determined by the Reynolds number (Re), as listed in Table 9.1. The Reynolds number is defined as

$$\text{Reynolds number (Re)} = \frac{\rho U d}{\eta} \tag{9.5}$$

where P is the density of the coolant, U is the averaged velocity of the coolant, d is the diameter of the cooling channel, and η is the dynamic viscosity of the coolant.

Table 9.1: Coolant flow types and corresponding Reynolds number ranges

Reynolds Number (Re)	Type of Flow
4,000 < Re	Turbulent Flow
2,300 < Re < 4,000	Transition Flow
100 < Re < 2,300	Laminar Flow
Re < 100	Stagnated Flow

9.5 Design Rules

9.5.1 Mold Cooling Design Considerations

The design rules presented here provide some guidelines for attaining proper and efficient mold cooling. Cooling channels should be of standard sizes in order to use standard machine tools, standard fittings, and quick disconnects. Based on the part thickness and volume, the mold designer needs to determine the following design variables when designing a cooling system:

- Location and size of channels of cooling channels
- Type of cooling channels (see Section 9.2)
- Layout and connection of cooling channels (see Section 9.2)
- Length of cooling-channel circuits (see Section 9.2)
- Flow rate and heat transfer of coolant

9.5.2 Location and Size of Channels

9.5.2.1 Part Thickness

To maintain an economically acceptable cooling time, excessive part wall thickness should be avoided. Required cooling time increases rapidly with wall thickness. This calculation is shown in Section 9.4.1. Part thickness should be as uniform as possible, as shown in Figure 9.15.

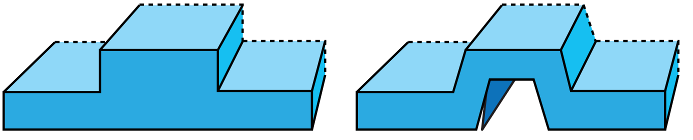


Figure 9.15 An alternative design can be used to maintain uniform part thickness.

9.5.2.2 Cooling Channel Location and Size

The best location for cooling channels is in the blocks that contain the mold cavity and core. Placing the cooling channels outside the cavity or core block will cool the mold poorly.

A primary goal when determining the size and location of cooling channels is mold surface temperature uniformity. The mold surface is the interface between the part and mold. The cooling channel depth and pitch (as shown in Figure 9.16) and the thermal conductivity of the mold material used all have a significant impact on the mold surface temperature distribution.

Figure 9.17 summarizes four examples of cooling channel configurations in P20 mold steel showing how the mold surface changes as the water line pitch and depth change. Figure 9.18 shows the location on the part where the temperatures were measured. In this example, the pitch of the cooling channels is 2.5 times the water line diameter or 2.5D with the diameter of 11.11mm (7/16 in). The coolant temperature was set at 30ºC and the cycle time was fixed at 17 seconds. With a depth of 1.0D and a pitch of 2.5D the mold surface temperature is fairly uniform, with the temperature difference of about 1ºC with an average temperature slightly less than 40ºC or 10ºC higher than the coolant temperature.

When the pitch is increased to 10D with the same 1D depth, the mold surface temperature difference increases to about 25ºC with an average of about 56ºC. The recommended starting point for depth and pitch spacing is 2.5D for the pitch and depth. With this configuration, the mold surface temperature difference is nearly uniform, but the difference between coolant temperature and the average mold surface temperature increases to just over 20ºC.

Finally, if the depth is increased to 5D and the pitch to 10D, the mold surface temperature is uniform within 2ºC, but the average temperature is 46ºC hotter than the coolant temperature.

As the thermal conductivity of the mold material changes, so does the temperature uniformity and temperature difference. In steels, the thermal conductivity ranges from about 10 W/mºC to about 40 W/mºC. Within this range of thermal conductivities, there is little significant difference in the uniformity of the temperature difference across the part, but there is a significant difference between the coolant temperature and the part. As thermal conductivity of the mold material increases, the spacing of depth and pitch become less important. Copper alloys have conductivities starting at approximately 60 W/m°C up to approximately 270 W/m°C. The mold surface temperature becomes more uniform and the difference between the coolant and mold surface becomes lower.

The temperature difference between the coolant and mold surface must be taken into account in both the design of the mold's cooling system and during production.

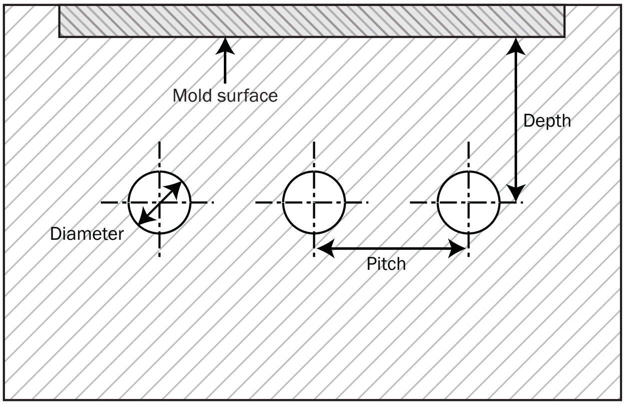


Figure 9.16 Typical dimensions for cooling channel diameter

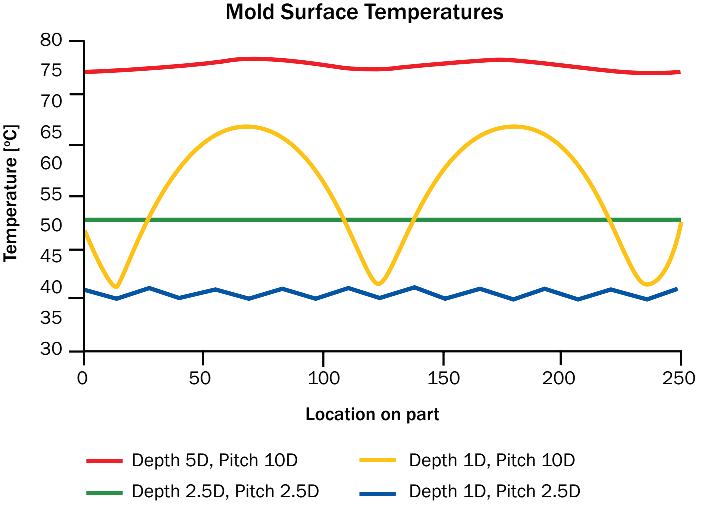


Figure 9.17 Mold surface temperature variations for different water line depth and pitch combinations with P20 mold steel

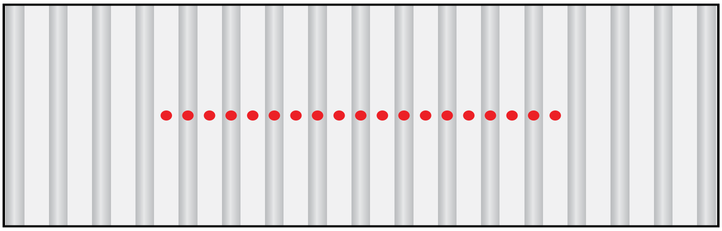


Figure 9.18 Red dots indicate the locations of temperature measurement, which are located directly above the coolant lines and mid-way between the coolant lines

9.5.3 Flow Rate and Heat Transfer

9.5.3.1 Temperature Difference on the Part

Keep the temperature difference on opposite sides of the part to a minimum; the mold surface temperature difference should not exceed 10°C (18°F) for parts that require tight tolerance.

9.5.3.2 Temperature Difference of the Coolant

In general, the temperature difference of the coolant between the inlet and the exit should be within 5°C (9°F) for general-purpose molds and 3°C (5°F)for precision molds. For large molds, more than one cooling channel series may be required to assure uniform coolant temperature and, thus, uniform mold cooling.

9.5.3.3 Heat Transfer of Coolant Flow

The effect of heat transfer increases as the flow of coolant changes from laminar flow to turbulent flow. For laminar flow, heat can be transferred only by means of heat conduction from layer to layer. In turbulent flow, however, the mass transfer in the radial direction enables the heat to be transferred by both conduction and convection. As a result, the efficiency increases dramatically. Figure 9.19 illustrates this concept.

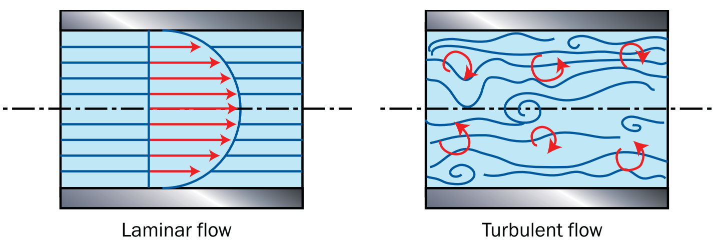


Figure 9.19 Laminar flow and turbulent flow

Once turbulence is achieved, the increase of heat transfer will diminish as the coolant flow becomes greater; therefore, there is no need to increase the coolant flow rate when the Reynolds number exceeds 10,000 to 20,000. Otherwise, the small, marginal improvement in heat transfer will be offset by the higher pressure drop across the cooling channels, along with more pumping expense.

Figure 9.20 below illustrates that once the flow becomes turbulent, a higher coolant flow rate brings diminishing returns in improving the heat flow rate or cooling time, while the pressure drop and pumping expenses are drastically increasing.

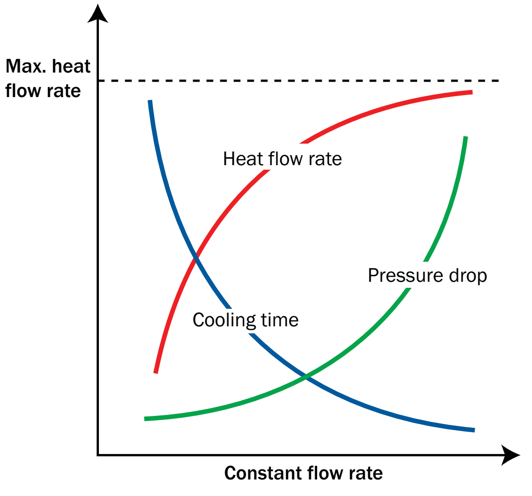


Figure 9.20 The relationship of heat flow rate and coolant flow rate.

It is important to make sure that the coolant reaches turbulent flow everywhere in the cooling system; a Moldflow cooling analysis can help identify and correct problems such as stagnated cooling channels, bypassed cooling channels, and high pressure drops in some cooling circuits

9.5.3.4 Air Gaps

A layer of air can impair the transfer of heat effectively. Therefore, take steps to eliminate any air gaps between the mold insert and molding plates, as well as any air pockets in the cooling channels.

9.6 Using Moldflow for Cooling System Design

This example uses a Moldflow cooling analysis to examine how process conditions, mold material, and part thickness affect the required cycle time.

9.6.1 Example Setup

Our goal is for the molded part to cool sufficiently for ejection without permanent deformation. To achieve this, Moldflow calculates the cooling time as the time required to cool 90% of the part volume to the ejection temperature specified by the user.

9.6.1.1 Cycle Time

The total cycle time is the sum of:

- Contact time

  The contact time is defined as the sum of:

  - Fill time
  - Packing time
  - Cooling time
- Mold opening time

9.6.1.2 Part Geometry

The molded part in this example is a simple plate measuring 200 mm x 150 mm x 2.5 mm.

9.6.1.3 Variables Used

For this example, the varying parameters are:

- Melt temperature
- Coolant temperature
- Ejection temperature
- Mold material
- Part thickness

9.6.1.4 Cooling Channels

The cooling circuits below the cavity are the mirror image of those above the cavity. They measure:

Diameter = 10 mm

Pitch = 2.5D or 25 mm

Depth = 2.5D or 25 mm

Figure 9.21 shows the configuration of the cavity cooling channels, a typical plot of the coolant temperature, and the cavity geometry of the plate. The core cooling channels have the same configuration as the cavity. The flow rate was set so the coolant temperature rise within the circuit is well within acceptable limits.

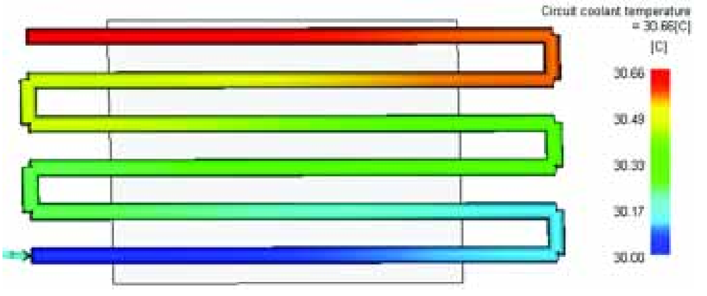


Figure 9.21 Circuit coolant temperature

9.6.2 Cycle Time Determined by Design and Processing Parameters

Table 10 gives a summary of the design and processing parameters used by the Moldflow cooling analysis and the predicted cycle times. Case 2 is the benchmark case. Figure 9.22 graphs the cycle times predicted. The mold wall temperature distribution of Case 2 is plotted in Figure 9.23.

9.6.2.1 Melt and Coolant Temperature

Higher melt and coolant temperatures require longer cycle times. The higher melt temperature (Case 3) increases the cycle time 1.5 seconds. However, increasing the coolant temperature 10ºC increased the cycle time 8.3 seconds. Cycle time is most influenced by changes in mold temperature.

9.6.2.2 Thermal Properties

The ejection temperature represents the temperature at which the part is cool enough to withstand ejection forces. In Case 6, when the ejection temperature was lowered so it had the same effect as raising the coolant temperature, the cycle time went up. The smaller the difference between the coolant temperate and the ejection temperature, the higher the cycle time will be.

As the mold material's thermal conductivity increases, the temperature gradient through the mold material decreases, reducing the mold surface temperature and therefore the cycle time. The stainless steel used has a thermal conductivity of 25 W/mºC and the copper alloy has a thermal conductivity of 250 W/mºC. Therefore, the cycle time did not increase much for the stainless steel (Case 10) and the copper alloy dropped a bit more (Case 11).

9.6.2.3 Part Thickness

The effect of part thickness on the cooling time can be seen in Cases 2, 8, and 9. Reducing the wall thickness has the most influence in reducing the cycle time for the part. This is clear evidence that thick sections of a part design should be removed.

9.6.2.4 Temperature Distribution

The mold surface temperature distribution shown in Figure 9.23 has been modified slightly to highlight temperature change. The temperature range on the part was slightly below 10ºC but was set to exactly 10ºC. The number of colors was reduced to five and banded so each color represents 2ºC. The majority of the part is within 2ºC For most parts, the temperature distribution should be within +/- 10ºC (18ºF) from the nominal mold temperature.

Table 10: Parameters used for Moldflow cooling analysis to predict cycle times

Case	Melt Temp (ºC)	Coolant Temp (ºC)	Ejection Temp (ºC)	Part Thickness (mm)	Mold Material	Predicted Cycle Time (sec)
1	210	30	80	2.5	P-20	20.5
2	230	30	80	2.5	P-20	21.6
3	250	30	80	2.5	P-20	23.1
4	230	20	80	2.5	P-20	20.1
5	230	40	80	2.5	P-20	29.9
6	230	30	60	2.5	P-20	29.8
7	230	30	100	2.5	P-20	17.3
8	230	30	80	2.0	P-20	15.0
9	230	30	80	3.0	P-20	29.4
10	230	30	80	2.5	420 SS	22.3
11	230	30	80	2.5	High TC Copper	18.4

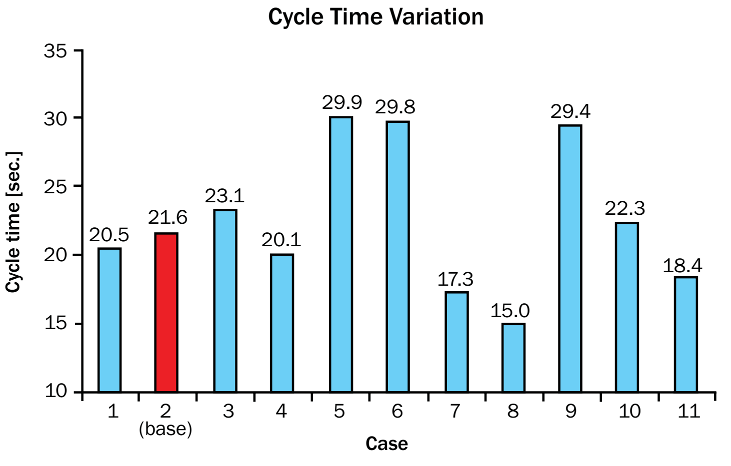


Figure 9.22 Graph of cycle time variation

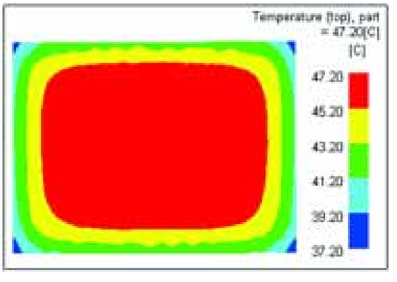


Figure 9.23 Mold-wall temperature distribution for Case 2

10 Shrinkage and Warpage

- Injection molding and shrinkage
- Basic causes of shrinkage and warpage
- Designing accurate parts considering warpage

10.1 Injection Molding and Shrinkage

In this section the relationship between processing and shrinkage is considered. In particular, the effect of packing pressure on shrinkage is described.

10.1.1 What Are Shrinkage and Warpage?

Part shrinkage may be thought of as a geometric reduction in the size of the part. If the shrinkage is uniform, the part does not deform and change its shape, it simply becomes smaller.

Warpage results when shrinkage is not uniform. If regions of the part shrink unequally, stresses are created within the part which, depending on part stiffness, may cause the part to deform or change shape. In the long term parts can even crack.

10.1.2 Shrinkage and Machine Settings

All molders know that shrinkage and consequently warpage is affected by processing conditions. Figure 10.1 shows some of the classic relationships between machine settings and shrinkage, also shown is the effect of wall thickness. These curves apply only to a particular mold and material combination. It is clear from Figure 10.1 that the final shrinkage of a component is a complex function of machine settings. Nevertheless, a major factor is the pressure and time history of the material as it fills, packs, and cools in the mold.

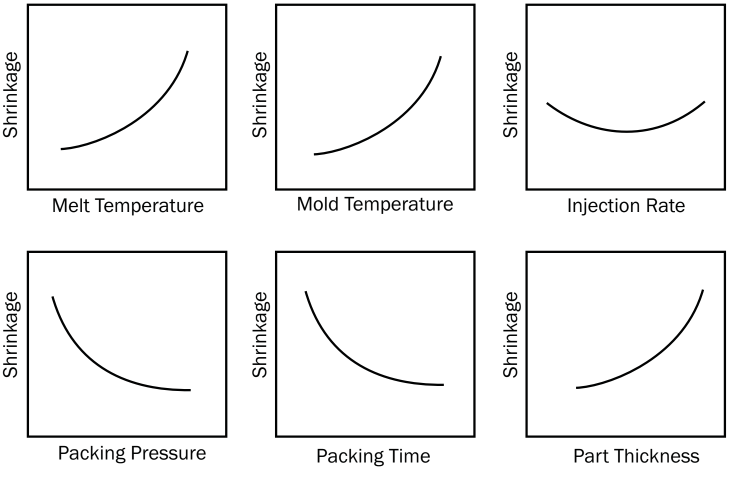


Figure 10.1 Effect of machine settings on shrinkage

10.1.3 Mold Filling and Packing

Plastic melts are very compressible at the pressures used in injection molding. As the ram moves forward, the material in the barrel is compressed so that the flow rate in the cavity is less than indicated by the ram movement. As the ram slows down, the plastic expands under pressure. Melt compressibility causes a smooth transition from mold filling to packing.

The molding process is frequently divided into two phases. Injection molders will commonly talk about the filling and packing stages because this corresponds to machine settings. Experiments on an instrumented mold show this concept is far from the truth. Figure 10.2 illustrates a simple mold with pressure transducers PT1, PT2, and PT3 positioned as shown. The lines labeled PT1, PT2, and PT3 show the pressures recorded by these transducers during filling of the mold.

Because of the compressibility of plastic, there is a time delay between ram displacement and plastic movement. This actual switch from filling to packing on the machine usually occurs before the cavity is filled (see Figure 10.2) and the final stages of filling occur by expansion of the pressurized material.

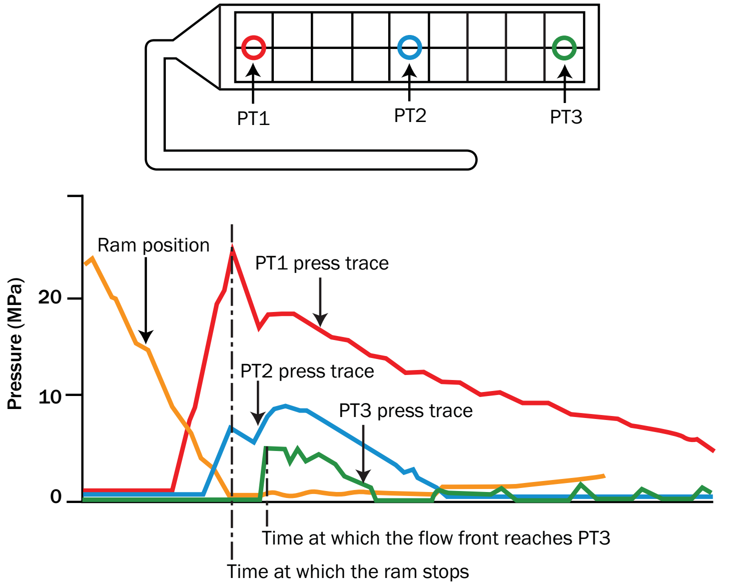


Figure 10.2 Pressure traces for a simple molding

10.1.4 How Pressure and Time Affect Shrinkage

The magnitude of pressure and the time for which pressure is applied greatly affect the shrinkage of material in the cavity. The actual pressure to which the material is subjected is determined not only by machine settings, but also by the viscosity of the material and the geometry of the cavity. Although a complicated matter, it is possible to restrict attention to two important regions: close to the gate and at the end of flow.

10.1.4.1 Shrinkage near the Gate

Areas near the gate are easier to pressurize (and depressurize) than areas at the end of flow and generally the relationship between pressure, time and shrinkage is simple.

High packing pressure gives lower shrinkages as long as the pressure is kept on until the gate has frozen. In this case the shrinkage around the gate will generally be lower than that at the end of flow.

If the packing pressure is not held on until the gate or runner system has frozen, then the pressure in the cavity will cause plastic to reverse flow back into the runner system. This can result in a higher shrinkage around the gate area than in the rest of the cavity.

10.1.4.2 Shrinkage at the End of Flow

Pressure has to be transmitted through the plastic to reach the extremities of the cavity. Cavity geometry, viscosity, and the time the melt channel in both the feed system and cavity remain open determine how well pressure is transmitted.

A high packing pressure results in a high initial flow as the pressure is quickly distributed throughout the cavity. Once the cavity is pressurized, the flow into the cavity will result from the contraction of the material and may be very slow in comparison with the initial flow. In other words there will be a high initial flow followed by a very slow flow.

A low packing pressure may give the opposite effect. Initially the flow rate will be much smaller than with the high pressure so the frozen layer will grow quickly. However as the material cools the volumetric change (from high to low temperature) is much greater at low pressures so the flow rate due to compensation will be greater than for the higher pressure.

High packing pressures do not automatically mean that there will be less shrinkage at the end of flow. This is because the plastic will freeze off in the upstream section earlier in the cycle, thus preventing the pressure packing out the area at the end of flow.

10.1.5 Thermally Unstable Flow

Plastic flow is self-reinforcing, that is, flow will carry heat into an area thereby maintaining flow. This was illustrated in Chapter 1. A disk with a thick outer rim was packed out to give a high compensating flow to the thick outer rim. The plastic does not flow as a thin disk but forms a series of flow channels that are self-reinforcing, maintaining plastic temperature and heating the mold, while other areas with low flows freeze off early in the packing phase.

The flow channels will be filled with highly orientated material that cools off at a later time than the remainder of the part. They act as tension members that will cause warping.

Two important applications of this effect occur opposite the sprue and at corners. Plastics are not simply viscous materials but have certain mechanical strength. As the plastic melt changes direction at the sprue, some force is required to physically deform the material as the direction of flow changes. This force comes from the face opposite the sprue and results in a highly asymmetric flow pattern. A similar effect occurs at corners where a slight temperature difference or elastic effects will initiate asymmetric flow.

Very small mold temperature variations that have virtually no effect in the filling stage will have a major effect in the packing stage. The position of cooling lines can dramatically affect packing stage flow. Once established, these flow patterns will not just be maintained but will continue to self-reinforce in the later stages of packing.

10.2 Basic Causes of Shrinkage and Warpage

This section describes the main causes of shrinkage and warpage. Instead of relating shrinkage to processing parameters, we consider some fundamental factors that affect shrinkage. These factors are volumetric shrinkage, crystalline content, stress relaxation and orientation.

Describing shrinkage and warpage in terms of these variables is preferable to using machine parameters, as the relationships of the latter to shrinkage are too complex to be used as design criteria.

10.2.1 Causes of Shrinkage

Shrinkage of plastic components is driven by the volumetric change of the material as it cools from the melt state to solid. Despite the apparent simplicity of this statement, it is important to note that the relationship between the volumetric shrinkage and the linear shrinkage of the component is affected by mold restraint, crystallinity and orientation. Warpage is caused by variations in shrinkage.

10.2.1.1 Volumetric Shrinkage

To understand shrinkage it is first necessary to appreciate just how large the volumetric shrinkage of plastics is.

All plastic materials have high volumetric shrinkages as they cool from the melt state to the solid. Without pressure, this is typically about 25%. Plastic parts cannot be made without, in some way, offsetting this large volumetric shrinkage. In injection molding, the application of high pressure can reduce this volumetric shrinkage, but by no means eliminate it.

Pressure: The relationship between pressure, volume, and temperature for a plastic material can be conveniently represented with a PVT diagram. Such a diagram relates specific volume (the inverse of density) to temperature and pressure. Figure 10.3 is an example of a PVT diagram. The specific volume is given by the surface over the plane defined by the pressure and temperature axes.

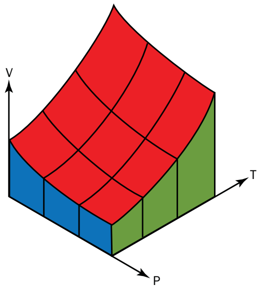


Figure 10.3 3D PVT diagram

PVT data for polymers usually is displayed as a projection onto the plane formed by the specific volume and temperature axes. Figure 10.4 shows this type of display for an amorphous and a semicrystalline material.

This diagram shows that normal injection molding pressures will only reduce volumetric shrinkage by around half. To see this, consider the points A, B, and C on Figure 10.4. Point A indicates the specific volume at room temperature and pressure, point B indicates the specific volume at a typical molding temperature, and Point C indicates the specific volume at a typical molding and packing pressure. The line going through point D is an extrapolated pressure line showing the pressure required to give zero shrinkage from the melt to the solid phase.

Such a pressure would be well in excess of that available on an injection-molding machine and clearly shows the impracticality of trying to eliminate shrinkage by the simple application of pressure alone.

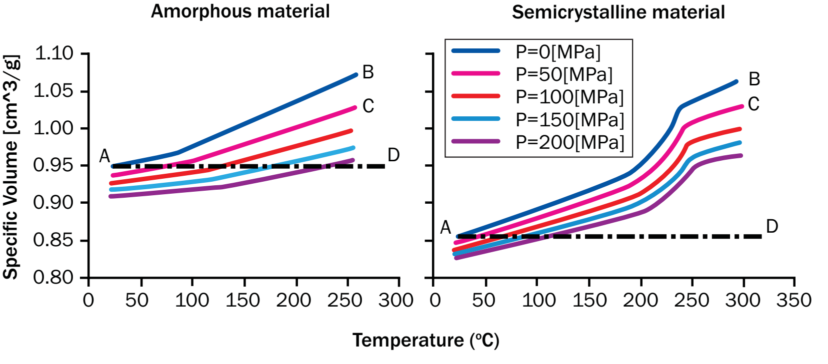


Figure 10.4 PVT diagrams for polymers

Crystallinity: PVT plots are usually measured at a constant temperature or very slow cooling rates. Under these conditions, the crystalline content will have reached equilibrium value. Volumetric shrinkage derived from PVT is therefore called equilibrium volumetric shrinkage.

Both cooling rate and orientation level will affect crystalline content. It is very difficult to obtain PVT data under conditions of fast cooling. In view of this, actual or net volumetric shrinkage is usually found by modifying equilibrium volumetric shrinkage with a mathematical model of crystallization kinetics.

10.2.1.2 Relationship between Linear and Volumetric Shrinkages

Linear shrinkage is driven by volumetric shrinkage, but there is not a one-to-one relationship. If the plastic were free to shrink in all directions isotropically, the linear shrinkage S_l would be approximately one third of the volumetric shrinkage S_v. In fact the exact relationship is

$$S_l = 1 - (1 - S_v)^{1/3} \tag{10.1}$$

Volumetric shrinkage for a given pressure, temperature and level of crystallinity will always be the same. However, the way volumetric shrinkage is divided into the three linear shrinkage components (thickness, parallel to flow, and perpendicular to flow) will vary.

The relationship between volumetric and linear shrinkages depends on stress relaxation and orientation.

Stress Relaxation: In practice, the two linear shrinkage components in the plane of the molding will have values much less than one third the volumetric shrinkage value. This is because the material is constrained in its own plane while within the cavity. It is however free to shrink in the thickness direction as shown in Figure 10.5.

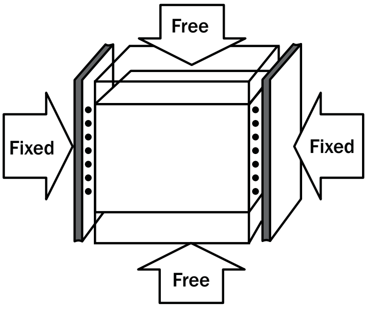


Figure 10.5 Effect of mold restraint

As the material tries to shrink in its own plane, stresses are created due to mold restraint. These stresses relax at a rate that depends on the relaxation characteristics of the material and the temperature-time history the part is subject to while constrained in the mold. These stresses will relax while the part is cooling, leading to permanent deformation of the part.

This is analogous to a stress relaxation experiment in which the material is stressed by applying a constant strain. Some of the stress will relax and result in permanent deformation while the residual stress will result in elastic deformation.

While in the mold, for each drop in temperature the material will receive a new stress input. At high temperatures most of this stress will simply relax while at lower temperatures a higher percentage of the stress will be retained elastically. If a cold part is simply left in the cavity longer, the effect on shrinkage will be quite small.

The cooling rate has a significant effect on the degree of relaxation. Raising mold temperature will reduce linear shrinkage relative to the volumetric shrinkage by allowing the material to relax (here, we ignore the additional crystalline content that may be produced by reducing the cooling rate). However, this may extend the cooling time.

Materials that relax slowly (materials with high resistance to creep) will be highly stressed in the cavity and so will spring off the mold and exhibit high linear shrinkage. Materials that relax quickly will tend to conform to the dimensions of the cavity and therefore have lower linear shrinkages. For a given volumetric shrinkage, materials that relax slowly will exhibit higher linear shrinkage than those with rapid relaxation characteristics.

Orientation: Orientation will cause the plastic to shrink by different amounts parallel and perpendicular to flow. Orientation of long stringy molecules is an easy concept to understand. Molecular orientation is initially generated by shear. At high temperatures, molecular mobility allows orientation to relax, so if shear stresses are removed the material will rapidly return to an unoriented state.

Orientation is locked in by the combination of freezing while shearing. Two factors influence the relationship between orientation and linear shrinkage. Usually oriented material will tend to relax, giving a higher shrinkage in the direction of flow than across the flow. For materials that crystallize, closer packing can occur perpendicular to flow, increasing shrinkage across the flow relative to shrinkage in the direction of flow. This effect is noticeable in materials prone to shear-induced crystallization.

It is important to note that for fiber-reinforced materials, orientation of the fibers has more effect than the molecular orientation. Also, the fiber orientation direction need not be in the direction of flow. For these materials, the above considerations do not apply.

Orientation and Shear-induced Crystallinity: Polypropylene is an interesting case. It will have high shrinkages parallel to flow at low levels of orientation because shear-induced crystallization effects dominate. Higher orientations result in higher shrinkages perpendicular to flow. This is illustrated in Figure 10.6.

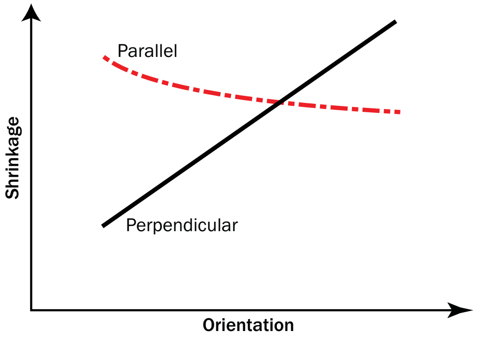


Figure 10.6 Parallel and perpendicular shrinkage of polypropylene

10.2.1.3 Summary

The main points to bear in mind from the above discussion are:

- Volumetric shrinkage is the driving force for linear shrinkage.
- Volumetric shrinkage is determined by pressure and temperature and, for crystalline materials, also by cooling rate.
- Volumetric shrinkage gives rise to three linear shrinkages in the directions of the thickness, parallel to flow and perpendicular to flow.
- The relationship between volumetric and linear shrinkages depends on the relaxation characteristics of the material and the effect of orientation.

10.2.2 Causes of Warpage

Warpage is caused by variations in shrinkage throughout the part. Shrinkage by itself does not cause warpage. A high uniform shrinkage will give a perfectly shaped part that is simply smaller.

Every point in the molding will have shrinkage parallel and perpendicular to flow. Computer simulation of shrinkage outputs three pieces of information for every element: a direction of material orientation and two shrinkage values (one in the direction of flow, one perpendicular). In considering how shrinkage causes warpage, it is convenient to define three types of shrinkage effects:

- Orientation effects
- Area shrinkage effects
- Differential cooling effects

10.2.2.1 Orientation Effects

Orientation effects arise from the difference between parallel and perpendicular shrinkages. These shrinkage differences tend to be useful on a local basis, that is, within some region of the part. The difference in shrinkage can be due to molecular or fiber orientation.

10.2.2.2 Area Shrinkage

For comparing shrinkages from region to region in the part, area shrinkage is a useful concept. Area shrinkage, also called differential shrinkage, is defined to be the change in area that occurs due to parallel and perpendicular shrinkage. Figure 10.7 illustrates this idea. Area shrinkages can be used to compare the difference in shrinkages between different regions of the molding such as near the gate and at the end of fill, while orientation effects can be used for comparing variations in shrinkage in different directions within a region.

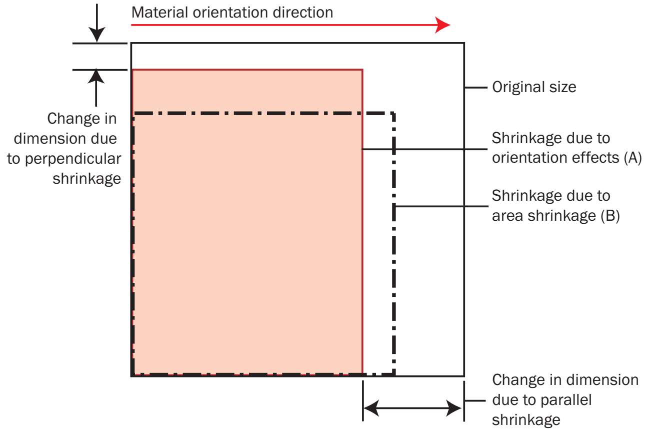


Figure 10.7 Definition of area shrinkage

10.2.2.3 Differential Cooling

Warpage can also be caused by variation in cooling. The most common example is from the difference in temperature on opposing mold faces. On removal from the mold the part may be perfectly flat, but as the part cools to a uniform temperature after ejection, the difference in contraction on the part's either side will set up a bending moment that results in warping. Warpage caused by differential cooling is very common in boxlike structures, such as in Figure 10.8. In this case the entire part is a "box," but the problem can occur in features on a large part as well. Generally, the problem is caused by the inside of the box being more difficult to cool than the outside, so the inside has a hotter mold temperature, which creates the temperature and shrinkage differential.

A similar effect may occur with thick and thin sections. It is well-known that thick and thin sections cause difficulty with crystalline materials because of difference in shrinkages. However, similar problems can still occur with amorphous material. The thick part may be hotter on ejection, and as it cools, the difference in shrinkage between the thick and thin section will set up internal stresses. The hot area will have had less constraint time in the mold than the cold area, giving rise to a difference in shrinkage.

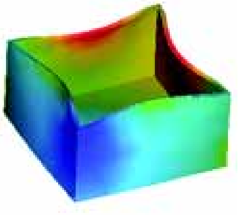


Figure 10.8 Warpage caused by differential cooling

10.2.3 Relating Orientation and Area Shrinkage to Warpage

Variations in shrinkage are the driving force for warpage. These are resisted by the geometric stiffness of the part. Stiff parts with high variation in shrinkage may not warp but will have higher internal stresses. If the part stiffness is reduced and the same shrinkages are applied, the part may warp but will have lower internal stresses.

Two important responses to orientation effects and area shrinkage can be illustrated with a center-gated disk. Though very simple, the disk exhibits behavior that is commonly seen in parts of more complex geometry.

Figure 10.9 illustrates two configurations that can result from molding. These are called dome- and saddle-type warpage. The causes of the warped shape are described in Table 10.1.

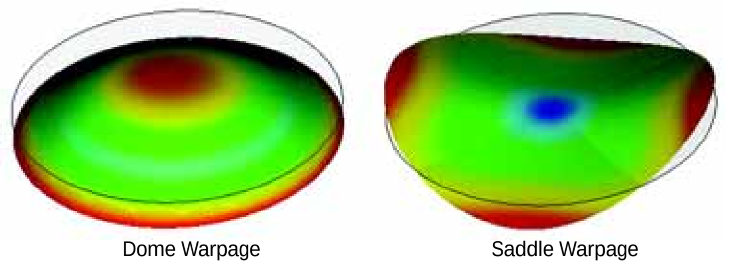


Figure 10.9 Dome- and saddle-type warpage for a disk

Table 10.1: Causes of dome- and saddle-type warpage

Warped shape	Cause of warpage
Dome	**Area shrinkage**. The outer region of the disk is higher than the area shrinkage of the inner region. This causes the circumference of the disk to decrease while the radius tries to maintain its value. By popping into the dome shape, the disk reduces its circumference while maintaining a similar radius. **Orientation effects**. Assuming the flow direction to be radial, if the perpendicular shrinkage is higher than the parallel shrinkage, the disk will also assume the domed shape.
Saddle	**Area shrinkage**. The the inner region has a higher shrinkage than that in the outer region. **Orientation effects**. The perpendicular shrinkage is less than the parallel shrinkage.

In summary, both dome and saddle shapes can be caused by either orientation or area shrinkage. Simple inspection of the part does not indicate which is causing the problem.

10.3 Designing Accurate Parts Considering Warpage

Part and mold designers can improve the dimensional stability of a product by considering:

- Material selection
- Wall thickness variation
- Gate position and runner dimensions
- Molding conditions
- Cooling line layout

In this section we discuss each of these areas.

10.3.1 Material Selection

Material selection is determined by many factors, one of which is shrinkage. In certain cases a material with low and uniform shrinkages will be specified to achieve dimensional accuracy. This inevitably involves a more expensive material. By applying the shrinkage and warpage design principles, it may be possible to use a more cost-effective material and achieve significant financial benefits.

An understanding of the basic shrinkage characteristics of different materials is useful in proper design. Shrinkage testing methods developed by Moldflow provide information on the shrinkage characteristics of different materials.

10.3.2 Wall Thickness Variation

Changing wall thickness will change both shrinkages and mechanical stiffness. Many traditional text books argue that parts should be designed with uniform wall sections. In practice a uniform wall section is not optimum and subtle variations can be used to advantage.

While most product designers will specify wall thickness, usually uniform, there is generally significant scope to vary local wall thicknesses to improve product performance. Changes in wall thickness will affect both orientation and area shrinkages. Increasing thickness will, in general, reduce orientation but increase area shrinkage.

10.3.2.1 Crystalline Materials

With crystalline materials, a change in crystalline content will normally dominate all other effects, so increasing thickness will increase area shrinkage.

10.3.2.2 Amorphous Materials

With amorphous materials, the dominating effect is a change in relaxation because of different cooling rates and the temperature at ejection. If the overall clamp close time, that is the duration for which the part is in the cavity, is increased, then area shrinkages will actually be reduced. If a short clamp close time is used, so the part is ejected hot, there will be less effective mold constraint, so shrinkages can be increased. This effect is particularly important when the part has wall thickness variations. If the part is ejected before the thicker regions have properly cooled, there will be an increased variation in shrinkages between thick and thin sections. This increases warpage stresses when the part has eventually cooled to room temperature.

10.3.2.3 Using Wall Thickness to Control Shrinkage near Gates

Analysis will often show that the area of maximum orientation is around the gate. This sets up differential orientation shrinkage that acts like tensile strings pulling the extremities of the molding toward the gate, and may result in part warpage.

Increasing the thickness around the gate will reduce the level of orientation and so reduce the component of warpage due to orientation effects. It will, however, often increase area shrinkage, which may increase warpage from differential area shrinkage.

If the area around the gate is increased in thickness and the area at the end of flow is also increased, uniform area shrinkage is maintained, and warpage will be reduced.

10.3.2.4 Using Wall Thickness to Control Orientation and Area Shrinkage

If the dominating cause of warpage is area shrinkage, then the part should be made thicker at the extremities than at the gate. On the other hand, if the dominating cause of warpage is orientation, then the part should be made thicker at the center and the extremities increased in thickness to reduce warpage components due to area shrinkage.

Because crystalline materials are more prone to area shrinkage, and amorphous materials are more prone to orientation, there is a trend to use a taper, thinner toward the gate with crystalline materials, and thinner away from the gate with amorphous materials.

The principles of varying wall section for both amorphous and crystalline polymers may be illustrated by referring to a simple box shown in Figure 10.10 (a). Consider first the case of saddle warping. This could be from either a high level of orientation or area shrinkage around the gate area. If the cause of warpage is orientation, as is likely to occur with an amorphous material, the solution is to increase the thickness around the gate as shown in Figure 10.10 (b).

If the cause is area shrinkage, this could be from either pressure distribution variation in the packing phase or variation in crystallinity (if relevant). The solution is to increase the thickness at the end of flow, possibly in conjunction with the use of flow leaders, seen in Figure 10.10 (c).

Note that the solution for orientation effects is opposite to that for area shrinkage. Accurate diagnosis of the cause is therefore critical.

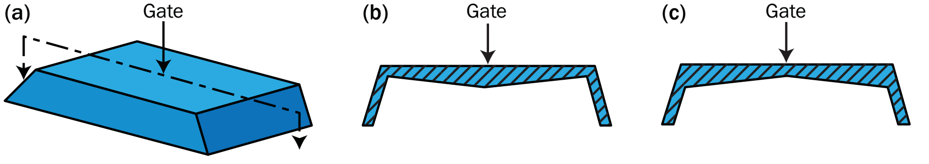


Figure 10.10 Using wall thickness to control warpage

10.3.3 Gate Position and Runner Dimensions

Design of the feed system is among the most critical factors in achieving dimensionally accurate parts. Moldflow design principles with respect to gate positioning and runner dimensioning relate to shrinkage and warpage.

The idea is to first try to position a single gate in the flow centroid of the molding (Figure 10.11 (a)). If a single gate is not practical, then the molding is mentally broken up into sub-moldings, gates positioned at the flow centroid of each sub-molding, and the runner system then dimensioned such that each sub-molding is filled at the same instant in time, as shown in Figure 10.11(b). The essence of this system is to avoid problems associated with overpacking, such as variation in shrinkage, part sticking in the cavity, etc.

These principles are valid when only a filling analysis is available but can be refined further to take advantage of packing, shrinkage, and warpage analysis capabilities.

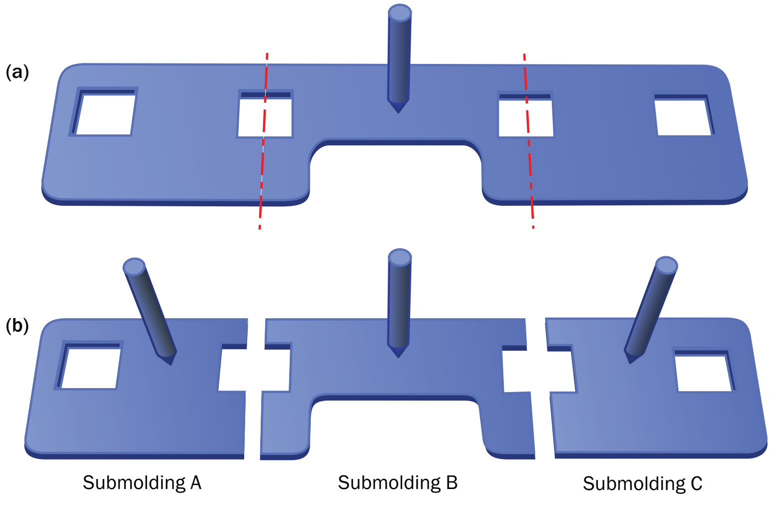


Figure 10.11 Gating and flow centroids

10.3.3.1 Design Criteria for Number and Position of Gates

The number and position of gates are determined using the criterion that the volumetric shrinkage at the end of flow should be close to the design value using the optimum packing pressure.

Minimum stresses, for a given shrinkage value, are obtained by having a high pressurization phase with a minimum compensating phase. In practice this means maximum packing pressure without flashing the mold. This is determined within the simulation by using a clamp tonnage ceiling.

It is possible to reduce shrinkage by using a low packing pressure that increases with time to maintain the critical flow rate, preventing freeze-off of the upstream flow path. This practice results in high levels of orientation and residual stress, so it should only be used if there are no other viable alternatives.

The gate(s) should be positioned to achieve both uniform and acceptable values of shrinkage at the end of all flow paths. In practice, this would be similar to the traditional criteria of filling all flow paths at the same instant in time; however, there may be some instances with thick and thin sections where it may be desirable to position the gate nearer thicker sections at the end of flow to achieve more uniform packing.

Gate positioning may be an important factor in reducing the effects of orientation. In some cases, changing the gate position is the only way of controlling orientation effects to produce a satisfactory design.

10.3.4 Molding Conditions

The basic principles of establishing molding conditions as developed in Chapter 2 remain valid. However, with the common use of injection profiles there is a need to revise these principles to take advantage of them.

Traditional design principles aim to have a uniform temperature over the cavity, that is, frictional heat is balanced against heat loss by conduction. In practice, most heat is generated around the gate areas, so the temperatures rise around the gate to a maximum and then drop off toward the end of flow. So the criterion is that the temperature at the end of flow should equal the temperature at the gate.

The aim in designing runner systems is to run the barrel temperature low to minimize degradation, and then use the runner system to generate frictional heat. This criterion does not consider problems such as jetting and other surface defects which tend to occur around the gate area. With programmed injection, the aims are to fill the cavity in the shortest possible time and to give a high temperature at the end of flow without causing jetting, overheating, or similar problems.

In addition to these molding constraints, there are also mechanical constraints arising from the flow rate and clamp tonnage capabilities of the machine. The ideal profile is generated by using the maximum injection rate and clamp tonnage of the machine as upper bounds, and then reducing the flow rate locally so that material constraints, such as maximum temperature, shear rate, and shear stress, are not exceeded.

It should be noted that the aim is to fill the mold in the shortest practical time and that by deliberately slowing the injection rate in certain key areas, normally around the gate and at the end of flow, a much shorter injection time can be achieved. In many cases, the available clamp tonnage will require a slower injection rate at the end of fill.

In the packing stage the aim is to bring the extremities of the cavity up to pressure without exceeding clamp tonnage limitations. A well-designed feed system should allow the cavity to be pressurized rapidly. The constant clamp tonnage criterion will usually result in the packing pressure decaying with time. This clamp tonnage is held constant until the plastic at the end of flow has frozen or until the required volumetric shrinkage has been achieved. In reality, these two criteria give the same results. The freeze front should then gradually progress back toward the gate. At each point in time, the packing pressure can then be adjusted to give the required volumetric shrinkage throughout the part.

10.3.4.1 Design Criteria for Volumetric Shrinkages

Volumetric shrinkages of molded material can vary widely. It is possible to produce negative volumetric shrinkages. In practice this means the part will actually expand on removal from the mold. This effect occurs if there is residual pressure inside the part at the end of the molding cycle. Typically negative volumetric shrinkages are associated with excessive packing pressure. At the other extreme, volumetric shrinkages can be over 15%.

Molding conditions that produce low or negative volumetric shrinkages are likely to be associated with high levels of orientation and hence high residual stress levels.

Very high volumetric shrinkages will usually result in excessive shrinkages in the thickness direction, which may be seen as poor surface finish or sink marks, even though shrinkages in the plane of the molding may still be within the normal range.

Under these conditions, mold constraint will be setting up very high stress levels in the part as the plastic is stretched and thinned within the cavity. Either of these extremes should be avoided.

10.3.4.2 Uniform and Acceptable Volumetric Shrinkage

The first step in the shrinkage design approach is to try to achieve a uniform and acceptable volumetric shrinkage throughout the part. Simply ensuring that the part will fill does not automatically guarantee that the shrinkages will be acceptable, particularly at the end of flow.

In Chapter 2, it was recommended that both pressure and temperature be limited. The pressure required to fill the cavity should be significantly less than the pressure available on the machine. The temperature at the end of flow should be adequately high, preferably close to the melt temperature at the entrance of the cavity. Experience has shown that these design principles have been reliable, if perhaps a little conservative. The aims can now be redefined in terms of volumetric shrinkage. The new criteria are to achieve both uniform and acceptable values of volumetric shrinkage over the cavity. Design values for acceptable volumetric shrinkage are determined as part of the material testing procedure.

10.3.4.3 Minimizing Sink Marks

These ideal conditions cannot be obtained in some parts, particularly those with thick and thin sections or very long flow paths. In general, thick sections should not be filled through thin sections, but if this is impossible to avoid, such as with bosses, an alternative molding strategy must be adopted.

Instead of starting off with the highest pressure available within clamp tonnage limitations, the pressure at the end of fill should be lowered so there is no backflow out of the cavity. The pressure is then held at a low value to allow the frozen layer to grow and then the pressure gradually raised to maintain a constant flow rate, which preserves a uniform frozen layer thickness. Pressure is raised until the maximum clamp tonnage criterion is reached and then the pressure decayed again using the same criteria of maintaining a constant volumetric shrinkage.

> This procedure, while effective for eliminating sink marks and increasing pack at the end of flow, gives rise to moldings with significant orientation and hence stress and should, therefore, only be used when there is no other alternative.

10.3.4.4 Melt Temperature

Increasing melt temperature may increase or decrease shrinkage. Typically, very low melt temperatures lead to relatively high shrinkage because the part cannot be adequately packed out. Increasing the temperature, up to a certain point, decreases the shrinkage because the viscosity is reduced and so packing pressure can be better distributed through the cavity. For a given packing pressure and time, further increases in melt temperature lead to higher shrinkages as the cavity is then filled with material of relatively low density.

10.3.4.5 The Effect of Molding Conditions on Orientation

The above criteria will minimize variation in volumetric shrinkage throughout the part, but does not directly consider the effect of orientation. Orientation is caused by the combination of shearing while freezing. In general, the conditions of filling the cavity as rapidly as possible within constraints will reduce orientation.

The majority of the orientation occurs in the packing phase, so if orientation is diagnosed as the major cause of warpage, then high melt temperatures (probably in conjunction with faster injection), coupled with higher but still uniform volumetric shrinkage, should be used.

10.3.4.6 Mold Temperature

Variations in mold temperature are a well-known cause of warpage. The classic bowing in of the sides of boxes is normally due to differences in temperature between the core and cavity.

If a part is warping due to poor cooling, the solution method is usually obvious, that is, repositioning of coolant lines, use of beryllium copper inserts, or heat pipes, increasing coolant flow, etc.

The aim when designing a cooling circuit is not necessarily to achieve a uniform mold temperature. The aim should be to achieve a uniform cooling time. This means that wherever practical, thicker areas of the part should have a lower mold temperature, while thinner areas should be deliberately run at a higher mold temperature, possibly by moving the cooling lines further away from the surface. A higher mold temperature will give more uniform shrinkage and lower orientation levels, but will require much longer cooling times. Raising mold temperature without increasing cooling times results in worse moldings because of the lower mold constraint time.

10.3.4.7 Packing Pressure

A packing pressure that is initially high and then reducies with time gives more uniform area shrinkage and decreases orientation. Generally such profiles are optimum for parts with little variation in wall thickness.

In the case of a thin section feeding a thicker region, the above approach will not be optimum. A high degree of pack can be obtained using a packing pressure that is initially low but increases with time. This minimizes sink marks in the thick region but causes high orientation. Often a better solution is obtained by a change in gating position.

10.3.5 Cooling Line Layout

It is a mistake to design cooling circuits based on getting cooling lines as close to the part wherever possible. Variations in mold temperature can be a very useful method for compensating for other sources of warpage. For example, centrally gated parts often have a tendency to twist or saddle warp due to a combination of high orientation at the gate and low shrinkages from overpacking. This can be offset by deliberately running the extremities of the mold very hot (to increase shrinkage), while running the gate area cold (to reduce shrinkage).

It should be noted that the heat load around the gate is much higher due to the longer flow time of the plastic and relatively high frictional heating. To achieve a temperature distribution in which the outside is hot and the gate region is cold may require a much distorted cooling circuit or, more realistically, twin circuits with their own temperature control.

Temperature differential on either side of the mold can be deliberately used to make a part deflect in a certain direction. A common example is a part with ribs that are thinner than the main surface. There is a natural tendency for the part to deflect away from the thinner ribs. By running the rib area hotter, the part can be deflected back to the required shape. In some cases, an insert of lower conductivity or containing a separate cooling circuit, can be used to form the ribs. This gives better temperature control of the rib temperature.

11 Moldflow Design Procedure

- Determine analysis objectives
- Moldflow analysis steps framework
- Using Moldflow to evaluate an initial design
- Using Moldflow to optimize the design

11.1 Determine Analysis Objectives

Every analyzed part has a different set of constraints in the form of objectives, restrictions, and guidelines. These constraints must be taken into consideration when doing an analysis. However, you do not want to over-restrict the analysis to the point where a solution cannot be found. Moldflow can help identify problems and solutions to those problems, but there must be flexibility in the design to allow for solutions not initially considered.

The objectives defined for a part are as varied as the parts that can be injection molded. However, below is a list of analysis objectives that you might have. Some are just flow analysis related; others will require cooling and warpage analysis.

- Will the part fill?
- What material will work best for my part with regard to fill properties (e.g. pressure, shear stress, and temperature distribution)?
- What processing conditions should be used to mold this part?
- Where should the gate be located?
- How many gates are required?
- Where will the weld lines be, and will they be of high quality?
- Will there be any air traps?
- How thick can the part be made?
- Is the flow balanced within the part with the fixed gate location?
- Are the ribs too thin to fill completely?
- Are the ribs so thick that they shrink too much?
- Can the part be packed out well enough?
- Will this snap fit break during use?

- Can the part be filled and packed in the press specified for the job?
- Are the runners balanced?
- What size do the runners need to be to balance the fill?
- Is the runner volume as small as it can be?
- Is the gate too big or too small?
- Can a 10-second cycle time be achieved?
- Will the part warp more than the 1.5 mm tolerance?
- Will the parts molded in a family tool assemble together?

✎ This is not a comprehensive list, but it gives you an idea of what can be done.

11.2 Moldflow Analysis Steps Framework

The exact steps taken to conduct an analysis on a part are as varied as the parts and problems to be solved. Results from an analysis will guide you to a solution path that will change depending on the problem and choices you make to fix the problem. Generally, there are several ways to solve a problem, some better than others. Following are the basic steps for conducting flow, cooling, and warpage analyses on a part.

11.2.1 The Whole Process

The procedure for analyzing a part from filling to warpage is charted in Figure 11.1. It follows the Moldflow design principles discussed in Chapter 4: the filling of the part, followed by the runners, cooling system, packing, then warpage. Each of these basic steps will be discussed in detail in the following sections.

11.2.2 Optimize Fill

Optimizing the filling of the part is the first major step in part optimization. This is the foundation for all other analysis work. The step therefore includes determining the objects as described above and preparing the model. The steps to optimize the fill are shown in Figure 11.2 and then described.

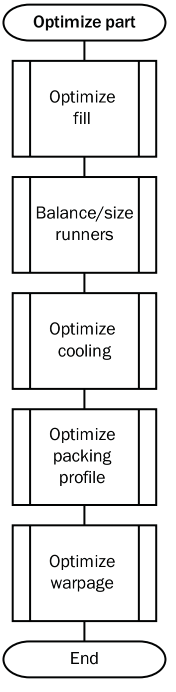


Figure 11.1 The basic steps for optimizing the part from fill to warpage

11.2.2.1 Prepare the Model

Preparing the model involves importing from a CAD system the geometry of the part and meshing it inside Moldflow, or importing a finite element mesh directly. This is discussed in Chapter 5.

11.2.2.2 Select the Material

To run an analysis in Moldflow, there are many material properties that have to be specifically tested for. What needs testing depends on the analysis to be preformed, but generally information about the material is needed in the following categories:

- Rheological properties
- PVT properties
- Thermal properties
- Filler properties
- Mechanical properties
- Shrinkage properties
- Recommended processing conditions

Moldflow has a very extensive database with the required properties needed for analysis. If a material is not found, the data can be obtained from Moldflow's material testing services, the material manufacturer, or other sources.

The material properties are a key input into the analysis. If poor material data is used, the results from the analysis will not be as reliable.

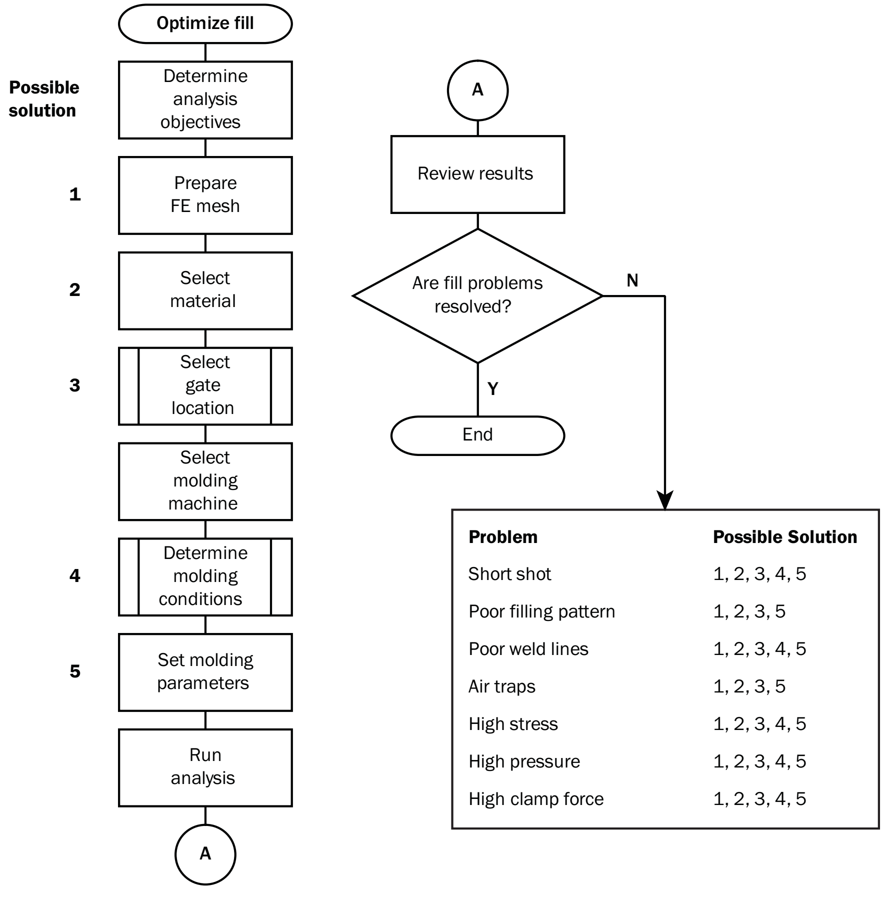


Figure 11.2 The steps required to optimize the filling of the part

11.2.2.3 Select Gate Location(s)

The number and location of the gates need to be determined. Many times this is not known and is one of the reasons Moldflow is being used. Some initial gate location must be defined. Figure 11.3 outlines the procedure. Additional information on gate locations is discussed in Chapter 7.

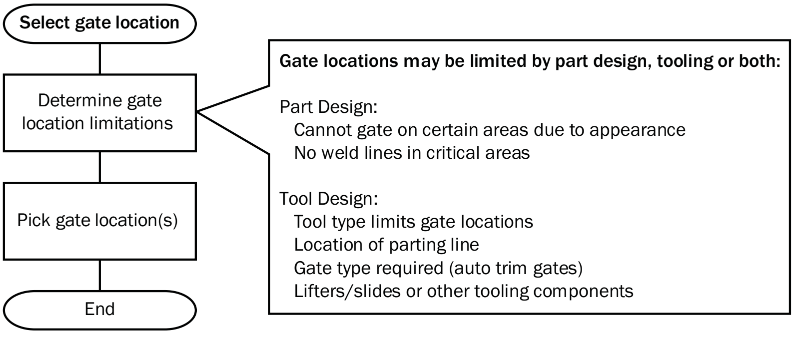


Figure 11.3 Selecting a gate

11.2.2.4 Select a Molding Machine

To run an analysis, some information about the molding machine being used must be defined: at least the injection pressure capacity and clamp tonnage capacity. Every analysis has a default molding machine defined that can be used in most cases. Moldflow also has an extensive database of molding machines so the specific machine can be used. For both the default and specific molding machines, all the parameters of that machine can be customized.

11.2.2.5 Determine the Molding Conditions

The molding conditions including the following must be defined:

- Mold temperature
- Melt temperature
- Injection time

These are the fundamental molding conditions needed for a fill or flow analysis. Optimizing these conditions is often a critical step for the part analysis and is outlined in Figure 11.4. Optimizing molding conditions often involves looking at the gate locations also. Discussion of molding conditions is found in Chapter 2.

11.2.2.6 Set Molding Parameters

Molding parameters include other inputs needed for the basic filling analysis. Many times default values are used, at least initially. Key parameters include the method and timing of the velocity to pressure switchover and the packing profile.

11.2.2.7 Run the Analysis

The analysis in this case is a filling analysis that only looks at the filling phase of the cycle. The analysis ends when the part is 100% full. The filling phase is the phase of the cycle where most problems will occur and/or be fixed.

Initial work on the part does not include the runner system, it only involves the part. This is discussed in Chapter 4.

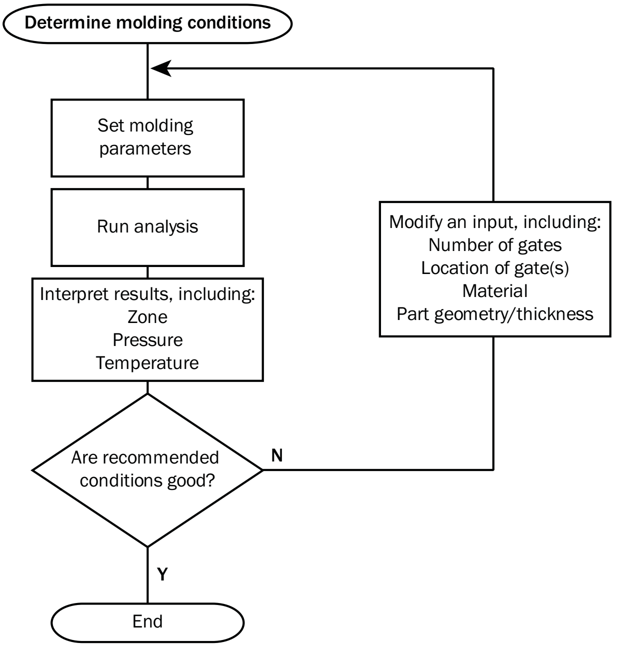


Figure 11.4 Determine molding conditions

11.2.2.8 Review the Results

When the filling analysis is done, the results of this analysis are reviewed to find any problems and to help determine how to proceed. There are a significant number of results that can be reviewed, but most are not used for every analysis. Which results are reviewed will depend on what the analysis objectives are and what problems are found.

11.2.2.9 Resolve Any Filling Problems

The results will highlight many problems. Figure 11.2 has a section listing just a small number of the potential problems a filling analysis can be used to find. The possible solution column lists different steps that may resolve the problem.

Resolving a filling problem is an iterative process. Never will just one fill analysis be enough. There will always be some question that needs to be answered requiring additional analysis. Depending on the nature of the problem and the proposed solutions, only two or three analyses may need to be run; in other cases many more iterations may be needed.

11.2.3 Balance and Size the Runners

After the filling of the part has been optimized, the sizing and balancing of the runner system needs to be done. Depending on the layout and complexity of the runner system, the runners simply may need to be sized to minimize volume, and other times the runners must be balanced to achieve the required filling pattern within and between parts. Chapter 8 discusses runner design and specifically sizing and balancing. The steps for balancing runners are shown in Figure 11.5.

11.2.3.1 Add Runners to the Part

Runners are added to the model once the part has been optimized. In most cases, the runner system is modeled within Moldflow and not imported.

> In some cases when the part has multiple gates and the part is not symmetric, the sizing of the runner system is integral to the part optimization. In these cases, runner systems are added to the part model before the part is completely optimized.

11.2.3.2 Determine Optimized Molding Conditions

During the part optimization process, the molding conditions for the part are optimized. Now that a runner system is being added, these conditions need to be modified to take into account the volume of the runners and the shear heat developed in the runners.

The volume of the runner needs to be accounted for because of the time required to fill the runners. Most of the time, the filling of the part is done by specifying an injection time. This time must be converted to flow rate by the following;

$$\text{Flow rate} = \frac{\text{Volume} \times \text{No. of Cavities}}{\text{Injection time}} \tag{11.1}$$

The volume is the volume of one cavity.

By using flow rate, whatever the volume of the runners will be, the fill time for the part itself will always be the optimum value. The total fill time will go up because of the volume of the runners.

The shear heat generated in the runners will change as the size of the runners change. However, the temperature entering the part should be within 5ºC of the optimized melt temperature determined in the filling analysis for the part. The shear heat in most runner systems will be more than 5ºC. It could be 30ºC or more. Run a filling analysis to determine the amount of shear heat. The temperature entering the sprue is then lowered by the amount of shear heat. This becomes a good starting point for further runner optimization.

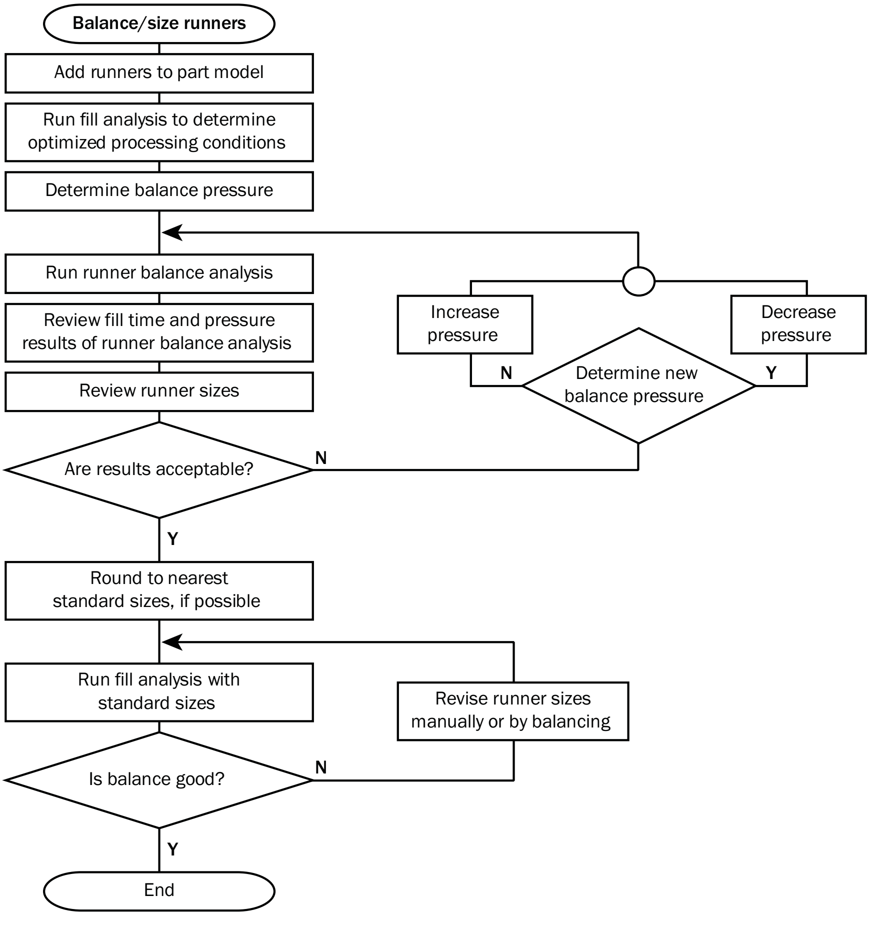


Figure 11.5 Balance and size the runners

11.2.3.3 Determine Balance Pressure

When the automatic runner balancing is done, a key input is the balance pressure. The analysis will change the size of runners so the pressure to fill the mold (runners and parts) is within the tolerance from the balance pressure. Even for geometrically balanced runner systems, the runner balance analysis can be used to downsize the runners to save volume and so the mold takes an acceptable amount of pressure.

When the runners are artificially balanced, the balance pressure becomes a critical input. If the pressure is too high, the runners will become too small, if the pressure is too low, the runners will become too large.

11.2.3.4 Review Runner Balance Results

Once the runner balance analysis is done, the fill analysis results can be reviewed. The balance is mostly determined by looking at the filling pattern and pressure results. If the results are not satisfactory, the balance analysis is done again with a revised balance pressure.

11.2.3.5 Review Runner Sizes

If the balance is acceptable, the sizes of the runners are viewed. The balance may be good, but the runner sizes may be too large or small. This would require another balance analysis with a new balance pressure.

11.2.3.6 Round Runner Sizes

Once the runner balance is acceptable, the sizes of the runners can be compared to standard sizes. The runner balance does not consider the size of the runner in relation to standards: it wants to make sure the parts fill uniformly.

If the sizes of the runners determined by the balance analysis are close to standard sizes, the runners can be changed to a standard size. The more the runners are changed the more “out of balance” the mold will become.

11.2.3.7 Run Analysis with Standard Sizes

If the runner sizes were manually changed to a standard size, a filling analysis should be run to ensure the filling of the part is still acceptable.

11.2.4 Optimize Cooling

Once the runner sizing/balancing is done, the cooling of the mold can be considered. Figure 11.6 shows the steps required to optimize cooling.

11.2.4.1 Determine Analysis Objectives

The cooling analysis objectives are the most important part of the process. What problems are to be investigated and/or solved? Generally, the mold surface temperature should be made as uniform as possible and the cycle time should be minimized.

11.2.4.2 Model Cooling Components

To run a cooling analysis, the cooling channels must be modeled. Possibly inserts can be modeled if high conductivity inserts are used. Other components can be modeled as well. Chapter 9 discusses design rules for placing cooling channels. Cooling components are normally modeled in Moldflow, but they can be imported as well.

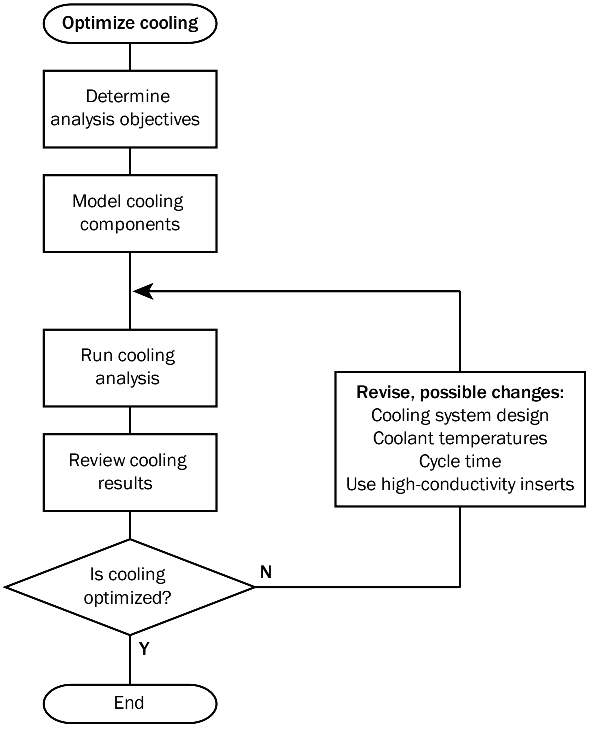


Figure 11.6 Optimize cooling

11.2.4.3 Review Cooling Results

The mold surface temperature is the most important result from cooling—the more uniform the results, the better. Other results from a cooling analysis help determine why there are problems with the temperature distribution.

11.2.4.4 Revise Inputs to the Cooling Analysis

When the cooling results are not acceptable, something needs to change. Generally, a cooling analysis is run to help place the coolant lines correctly and to determine coolant temperatures, flow rates, etc. These parameters are often revised in the process of optimizing the cooling.

11.2.5 Optimize the Packing Profile

Packing is best done after a cooling analysis because the packing and compensation phases are heat transfer dominated. When a packing analysis is conducted after cooling analysis, the tool's ability to extract heat from the part is accurately modeled. Figure 11.7 shows the steps involved in optimizing a packing profile.

The primary output from a packing analysis is the volumetric shrinkage. The amount and distribution of the volumetric shrinkage is critical for determining the linear shrinkage and warpage of the part. To optimize the volumetric shrinkage, the range of volumetric shrinkage is minimized.

11.2.5.1 Determine Initial Packing Pressure and Time

The maximum packing pressure that can be used is related to the machines clamp force and is determined by

$$P_{max} = \frac{\text{Machine Clamp force limit}}{\text{Total projected area of the shot}} \times \text{Unit conversion} \times 0.8 \tag{11.2}$$

where:

$$\begin{aligned} P_{max} &= \text{The maximum packing pressure that could be used.} \\ \text{Machine clamp force limit} &= \text{Tonnes(metric) or tons(US / English units)} \\ \text{Total projected area of the shot} &= \text{cm\^{}2 or inches\^{}2} \\ \text{Unit conversion} &= 100 \text{ for metric units, 2000 for english units} \\ 0.8 &= \text{Safety factor to only use 80\% of machine capacity} \end{aligned}$$

Often the maximum pressure that could be used based on the equation above is much higher than is needed. Packing pressurizing are generally 80 to 100% of the pressure required to fill the part. Packing pressures, however, can be much higher or lower however.

An initial estimation of packing pressure is often changed in the process of optimization.

11.2.5.2 Review Results

Volumetric shrinkage, pressure traces, and gate freeze time are the results most often reviewed to determine whether the shrinkage is acceptable or not. If the shrinkage is not acceptable, these results are used to determine how the packing profile must be changed to improve the volumetric shrinkage. To fully optimize the volumetric shrinkage, the process is iterative.

11.2.6 Optimize Warpage

Optimizing warpage, shown in Figure 11.8, is the last major step in part optimization. It encompasses all the steps previously discussed. The decisions made previously will influence the results here. In the process of reducing warpage, the solution may require going back to any of the previous steps defined earlier.

However, when design guidelines are properly followed, warpage will be minimized, and the process of warpage optimization becomes a validation process that previous work was well done.

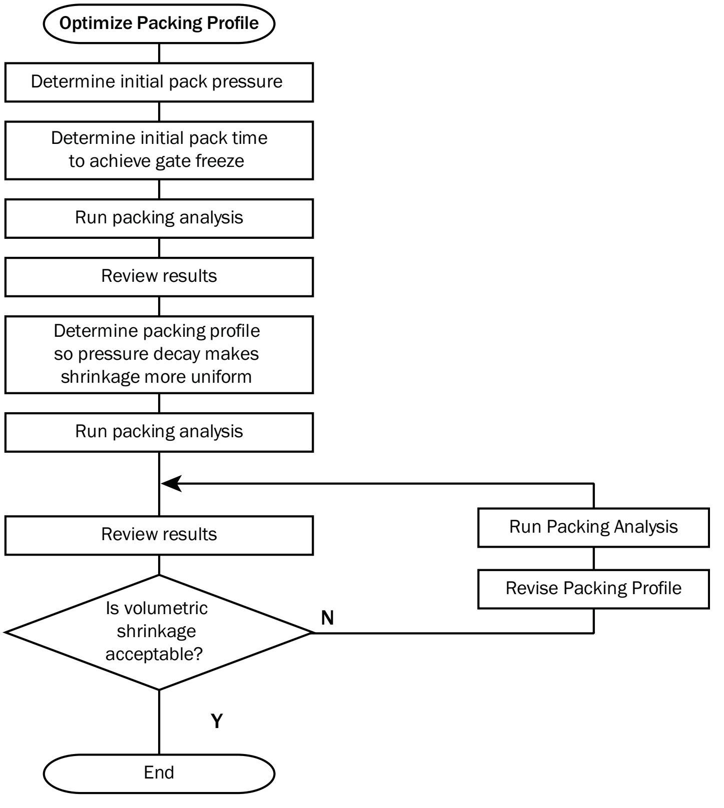


Figure 11.7 Optimizing the packing profile

11.2.6.1 How Warpage Is Defined

One of the most significant challenges when doing a warpage analysis is defining how warpage is measured on the molded part. This needs to be well understood so that results from the warpage analysis can be compared to how molded parts will be measured. Without this understanding, the warpage analysis will be improperly utilized.

11.2.6.2 Determine the Magnitude of Warpage

The first step is measuring how much the part has warped with the current processing conditions. The results from the warpage analysis are compared to the tolerances for the part to determine if the part is acceptable or not. When the parts don't meet the tolerances, the warpage must be reduced.

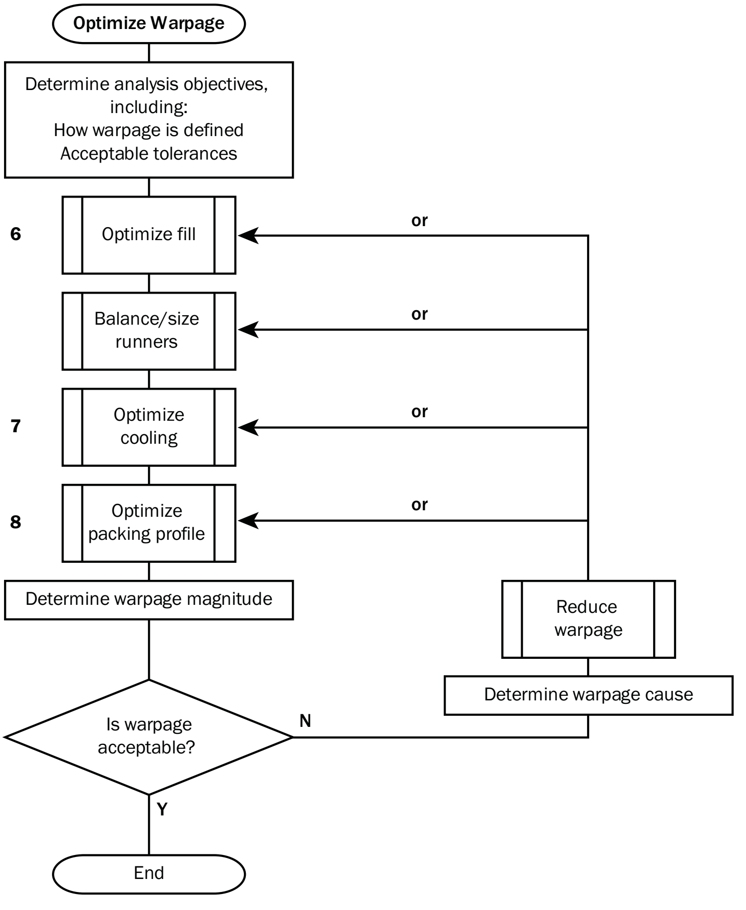


Figure 11.8 Optimize warpage

11.2.6.3 Determine the Cause of Warpage

If the part warps more than the tolerance allows, the warpage must be reduced. Chapter 10 discusses the complex interaction of part geometry, material, and processing parameters that lead to warpage. However, warpage can be broken down to three main causes as discussed in Section 10.2.2, including:

- Orientation effects
- Area shrinkage effects
- Differential cooling effects

Once these three main causes of the warpage can be determined, the warpage can be reduced.

11.2.6.4 Reduce Warpage

Reducing the amount of warpage to get it below the tolerance involves addressing the primary cause of warpage as shown in Figure 11.8. Fixing the primary cause of warpage can be done in many ways. The numbers beside possible ways to fix the warpage refer to steps numbered in Figure 11.9.

Often, reducing the amount of warpage is an iterative process. Several possible solutions could be found and evaluated to determine which one is most practical or economical. Many times it will take several iterations before the warpage is reduced enough.

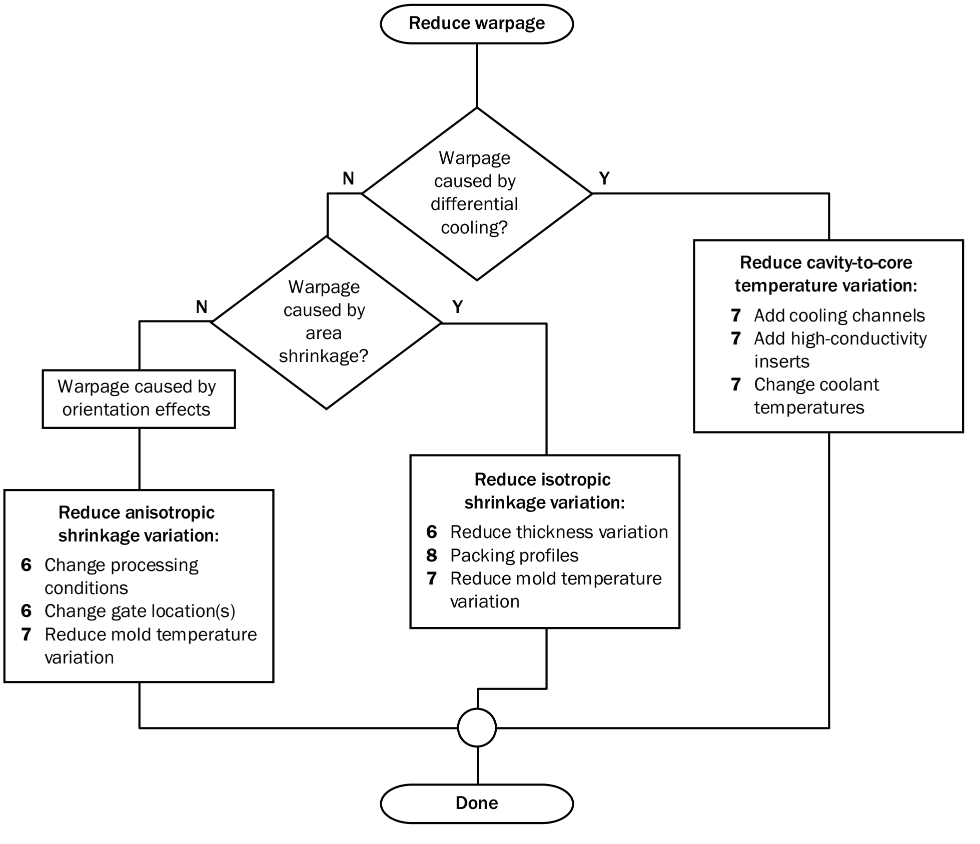


Figure 11.9 Reducing warpage

11.3 Using Moldflow to Evaluate an Initial Design

11.3.1 Description of this Example

The part for this example is a cover. The initial design, shown in Figure 11.10, is based on a previous year's design. The part has more rigorous specifications than in previous years. In this phase, the initial design will be evaluated to determine if further action is necessary.

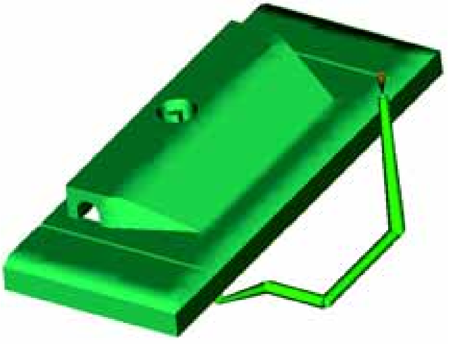


Figure 11.10 Original cover design

11.3.1.1 Specifications

The cover will be made in a two-plate, two-cavity tool, using a molding machine with a 100-tonne clamp force, and a 140 MPa pressure limit. Only one cavity is modeled because the second cavity is symmetric to the first cavity. The symmetry is accounted for by a technique called occurrence numbers. The material is a 33% glass-filled nylon 6. With the redesign for a new model year, the flatness of the bottom edge must be within 1.0 mm because of assembly requirements. The initial gate location is a submarine gate into a pin on the underside of the part. If necessary, other gate locations around the perimeter of the part may be used.

11.3.2 Molding Window

A molding window analysis was done on the cover, excluding the runner system. The results from the analysis are shown in Figure 11.11. They indicate that the processing window is wide with regard to mold and melt temperature ranges. The pressure to fill is well below the limit of the machine. The injection time range is about one second wide. Although it would be nice to have a wider range of injection times, a one-second range is very acceptable. The molding conditions picked are:

- Mold temperature:80ºC (176ºF)

- Melt temperature:280ºC (536ºF)
- Injection time:0.9 seconds

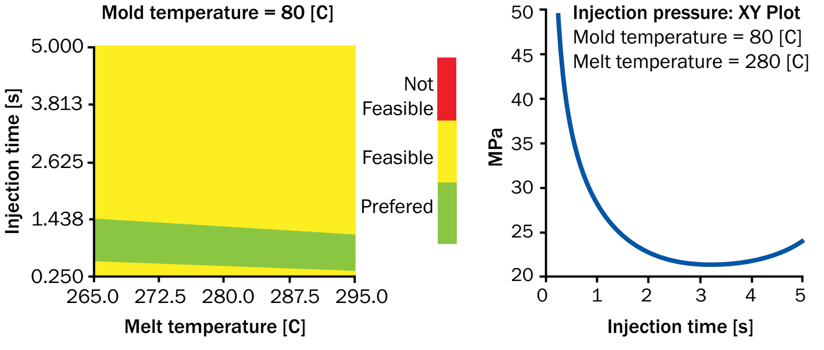


Figure 11.11 Molding window on the cover

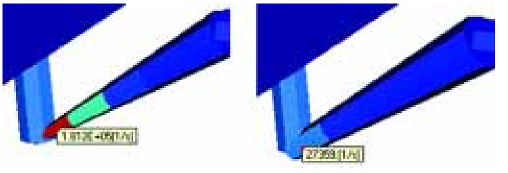
11.3.3 Filling Analysis

Because the gate location for the initial analysis was fixed from a previous design, the gate location cannot be optimized for now. The gate location does not produce a balanced fill pattern. The part to the left of the gate, as shown in Figure 11.12, is overpacked. In addition to being overpacked, this area has a lower temperature and thicker frozen layer. The flow front velocity is not uniform, and could be addressed with a velocity profile if necessary. Other results did not show any problems other than the ones discussed above.

Figure 11.12 Cover fill time plot

11.3.4 Gate and Runner Design

The original gating system has a submarine gate into a pin. The diameter of the gate was 1.0 mm. With the flow rate required to fill the part in 0.9 second, the shear rate in the gate is very high, over 181,000 1/sec., as shown in Figure 11.13. The shear rate limit for nylon is 60,000 1/sec. However, because of the glass fillers in the material, the shear rate should be well under the 60,000 1/sec. limit. The gate was opened up to 2.25 mm to reduce the shear rate to under 28,000 1/sec. For a glass-filled material, the shear rate could go a little lower, but it is not bad.



Original Gate Revised Gate

Figure 11.13 Cover gate shear rates

With the original gate and runner system, the shear heat in the runners was over 20ºC (36ºF). The original analysis used the optimized melt temperature as a starting point. Subsequent analyses lowered the melt temperature entering the sprue, so by the time the flow front got to the part the melt temperature would be about 280ºC (536ºF). The revised analysis lowered the melt temperature to 267ºC (513ºF), so the temperature entering the part was about the optimized temperature.

The cooling time of the feed system was compared to the part. The base of the sprue has the longest cooling time and is about 19 seconds, compared to the about four seconds for most of the part. However, there is a heavy boss on the cover that cools in about 12 seconds. The guideline says that the maximum runner cooling time should be no more than about two times that of the part. Considering there is a heavy section on the part that takes 12 seconds to cool, and the sprue cools in less than 24 seconds, the sprue cooling time is not unreasonable. Also, the sprue's orifice is the smallest standard commonly used, and it would not be practical to make it smaller. In addition, since the material is a glass-filled nylon, a smaller percentage of the cross section needs to be frozen for the part to be ejectable.

11.3.5 Cooling System Design

The cooling system for the cover consists of one circuit for the cavity and core, each having four lines. In the final design, the only thing that changed was a BeCu insert, which was added in the core to extract heat better. Figure 11.14 shows an example of the revised cooling system.

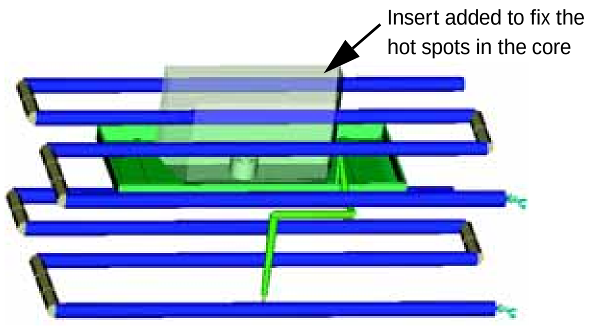


Figure 11.14 Revised cooling system

11.3.5.1 Mold Surface Temperature

The mold surface temperature of a part should be as uniform as possible. For semicrystalline materials this should be +/- 5°C from the target temperature, in this case 80°C (176°F). This is normally very difficult to achieve. The original design overcools much of the cavity, and the core runs very warm, up to 124°C (255°F). The main problem is the core is the narrow space between the boss and side wall. This narrow area adds 15°C to the core. A BeCu insert was used in the core to help extract the heat more efficiently. The core is now quite uniform in temperature. More than 50% of the surface area of the part is now within the tight +/- 5°C tolerance, and less than 10% is outside +/- 10°C tolerance, mostly to the cold side.

For both the original and revised designs, the coolant temperatures were adjusted to ensure the average mold surface temperature was close to the target temperature. In the case of the original design, the inlet temperatures were different for the cavity and core circuits. This is not always practical. With the revised design, the coolant temperatures were the same.

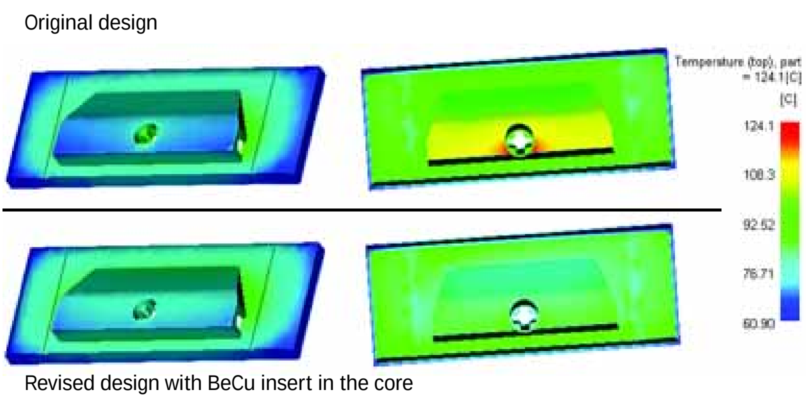


Figure 11.15 Mold surface temperature for the cover

The improvement in the temperature uniformity can also be seen in the temperature profile of the part. In Figure 11.16, two locations were tracked through the thickness of the part. One location (L1) is where the core runs very hot, and the other location is near the middle of the part (L2). The core side at both locations 20 to 35°C hotter in the original design than the revised design. Because the cavity side coolant line design was not modified, the temperatures on the cavity side are quite similar.

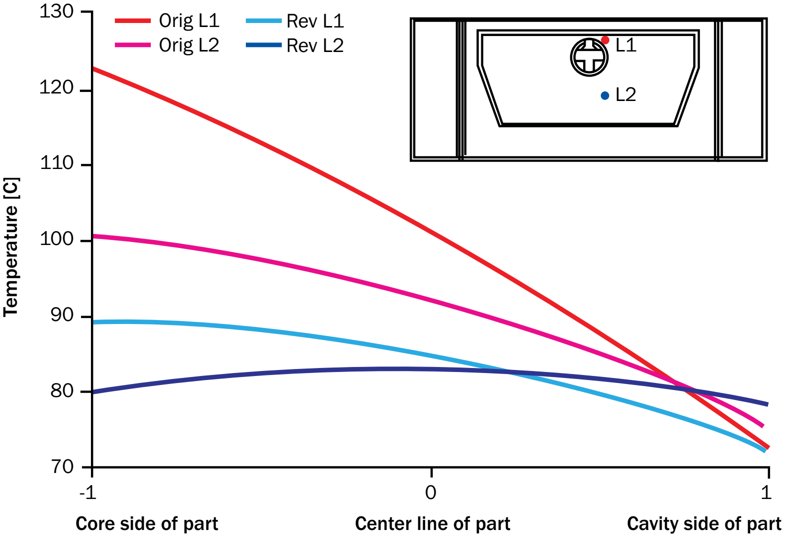


Figure 11.16 Temperature profile of the original and revised cooling designs at two locations on the part

11.3.6 Packing Analysis

The packing analysis was run to achieve a reasonable range of volumetric shrinkage without a significant amount of optimization. The material used is glass-filled, and the orientation of the fibers normally dictates the warpage. If the warpage is out of tolerance and the cause of warpage is differential shrinkage, then the packing can be optimized. The range of volumetric shrinkage is 3 to 5.1%, which is a good range.

11.3.7 Warpage Analysis

Warpage analysis, which requires input from a packing analysis, can now be run because the packing analysis has finished. The primary concern with the warpage analysis is the flatness of the bottom edge of the part. The edge must be flat within 1.0 mm (0.039 in). The warpage results were set up to make it easier to determine the flatness. Figure 11.17 shows the warpage of the bottom edge exaggerated 10 times to help show the change in shape and the magnitude of the warpage. Three of the four corners were anchored to form a reference plane to measure the deflection. The edge warped about 1.5 mm (0.059 in). This is 1.5 times the tolerance.

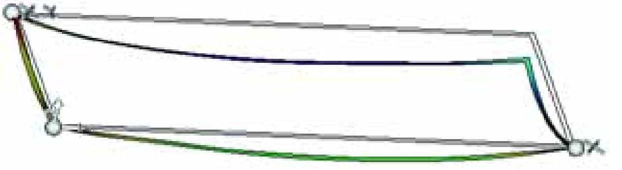


Figure 11.17 Bottom edge warpage of the cover's original design; the warpage magnitude is exaggerated 10 times to better show the change in shape

11.4 Using Moldflow to Optimize the Design

11.4.1 Determine the Cause of Warpage

Now that warpage analysis indicates the cover is well outside the flatness tolerance, the warpage must be fixed. Before the warpage can be fixed, the cause of warpage must be run to determine the cause.

For many parts, there are a combination of causes and offsetting effects. For example in some parts, the primary cause of warpage is differential shrinkage, but differential cooling is offsetting some of the differential shrinkage, so the total warpage is less than the differential shrinkage.

In the case of the cover, virtually all of the warpage is caused by orientation effects. Both differential cooling and differential shrinkage have slight offsetting effects.

11.4.2 Investigating Different Gate Locations

For fiber-filled materials, orientation effects could mean molecular orientation or fiber orientation. Fiber orientation dominates over molecular orientation. How fibers are oriented is very complex. Orientation can change through the thickness from highly oriented in the direction of flow, to random, to highly oriented across the direction of flow. Radial flow fronts tend to orient fibers across the flow direction, and shear rate tends to orient fibers in the direction to flow.

The gate location has a large influence on fiber orientation because of its influence on the shape of the flow front. Several different gating locations were investigated around the edge of the part.

Figure 11.18 shows how three different gate locations on the edge of the part compare to the original gate location. The gate location in the center of the long side make the warpage worse. With one or two gate locations on the narrow side of the part, the warpage is reduced, but not below the 1.0 mm tolerance. The different gate locations do not include runner systems, and the processing conditions are the same. The injection time could be optimized for the gate locations possibly improving the gate locations. However, other alternatives need to be investigated.

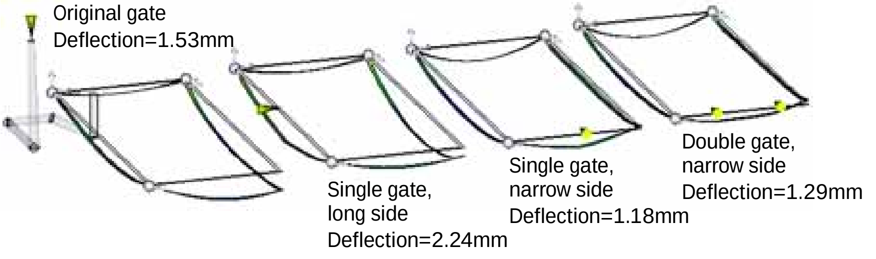


Figure 11.18 Cover comparing gate locations and deflections of the bottom edge of the part

Initially ruled out as a possibility, a gate location in the center of the part was tried. Figure 11.19 shows the gate location and resulting warpage of the part's edge. This shows that the warpage of the edge is now under the tolerance. By changing the gate location, the tool design needs to change from a two-plate tool with cold runners to a two-plate tool with a hot drop. Done early enough in the design phase, changing the tool configuration is not a significant issue, and the benefits can be easily shown using Moldflow.

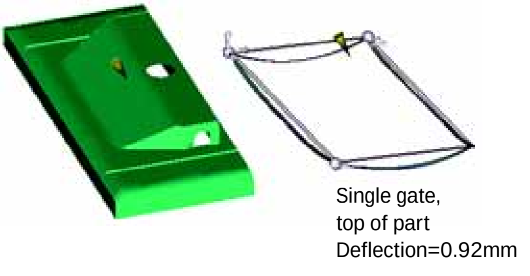


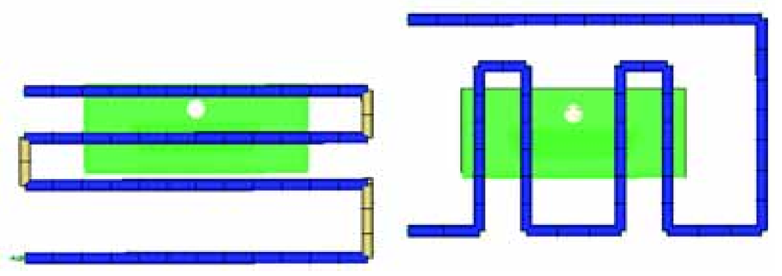
Figure 11.19 Cover deflection with a gate on the top of the part

11.4.3 Validating the Best Gate Location

The different gate locations were investigated without a runner system attached. Now that the gate location has been moved, the feed system for the optimized gate location has to be designed.

11.4.3.1 Re-design the Cavity Cooling Lines

Because the gate location was moved to the center of the part, the coolant line locations needed to be re-designed because of the hot drop location on the part, as shown in Figure 11.20. Primarily, the lines across the part were oriented in the y-direction rather than the x-direction.



Original cavity coolant lines Revised cavity coolant lines

Figure 11.20 Original and revised cavity coolant line locations

11.4.3.2 Design a Hot Drop

With the location of the gate set, another molding window analysis was done to validate the injection time. Because the flow length was shorter, the injection time should be a little faster. Even though the original 0.9 seconds will work, 0.7 seconds was found to be a bit better. The orifice for the hot drop was initially set at 2.0 mm. The first analysis indicated that the shear rate was nearly 80,000 1/sec., which is above the shear rate limit for the material. The warpage analysis indicated that the warpage was lower with the faster injection time.

The gate was opened to 2.5 mm. The flow rate was also increased to make the nominal injection time 0.6 seconds. The warpage dropped again. The shear rate was also lowered to 53,000 1/sec. This shear rate should be lower because of the glass fillers, but the orifice is larger than the nominal wall of the part. The hot drop vendor should be consulted to ensure the gate orifice can be made to the desired size without leaving a large mark on the part.

11.4.3.3 Warpage with the Runner System

Once the hot drop was sized, a packing analysis was done so the volumetric shrinkage was about the same as the analysis with the injection location directly on the part. With the changes in the cooling and filling the warpage was better than just the gate location on the part. A comparison of the original gate location and the final is shown in Figure 11.21.

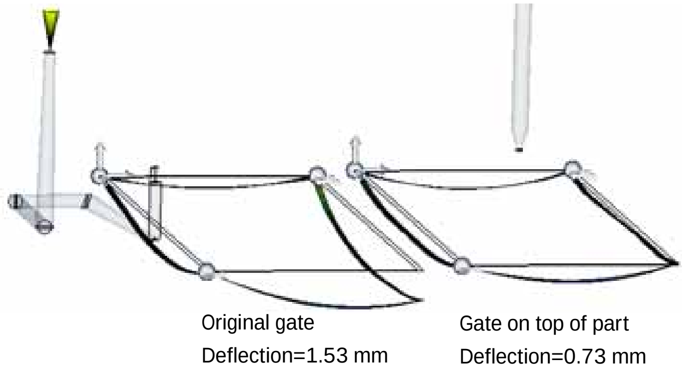


Figure 11.21 Warpage of bottom edge of the part with original and final gate locations

12 Part Defects

- Air traps
- Black specks and black streaks
- Brittleness
- Burn marks
- Delamination
- Dimensional variation
- Discoloration
- Fish eyes
- Flash
- Flow marks
- Hesitation
- Jetting
- Ripples
- Short shot
- Silver streaks
- Sink marks and voids
- Weld lines and meld lines

12.1 Air Traps

12.1.1 What Is an Air Trap?

An *air trap* is air caught inside the mold cavity. It becomes trapped by converging polymer melt fronts or because it failed to escape via the mold vents, or mold inserts, which also act as vents. Air-trap locations are usually in areas that fill last. Lack of vents or undersized vents in these last-to-fill areas are a common cause of air traps and the resulting defects. Another common cause is racetracking (the tendency of polymer melt to flow preferentially in thicker sections), caused by a large thickness ratio.

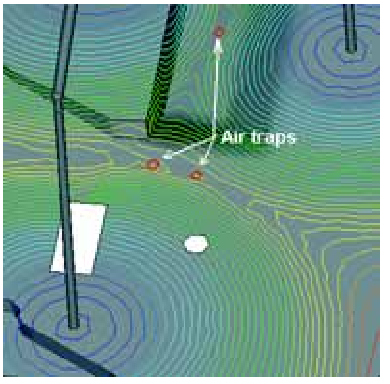


Figure 12.1 Air trap locations indicated by the air trap and fill time simulation results.

12.1.2 Problems Caused by Air Traps

Trapped air will result in voids and bubbles inside the molded part, a short shot (incomplete fill), or surface defects such as blemishes or burn marks.

12.1.3 Remedies

12.1.3.1 Alter the Part Design

Reducing the thickness ratio will minimize the racetracking effect of polymer melt.

12.1.3.2 Alter the Mold Design

Pay close attention to the proper placement of your vents. Place vents in the areas that fill last. Vents are typically positioned at discontinuities of mold material, such as at parting surfaces, between the insert and mold wall, at ejector pins, and at mold slides.

Re-design the gate and delivery system. Changing the delivery system can alter the filling pattern in such a way that the last-to-fill areas are located at the proper venting locations.

Make sure the vent size is large enough so that the air present in the cavity can escape during injection. Be careful, however, that the vent is not so large that it causes flashes in the vent. The recommended vent size is 0.025 mm (0.001 in) for crystalline polymers, and 0.038 mm (0.0015 in) for amorphous polymers.

12.1.3.3 Adjust the Molding Conditions

Reduce the injection speed. High injection speeds can lead to jetting, which causes air to become entrapped in the part. Lowering the injection speed will give the air displaced by the melt sufficient time to escape from the vents.

12.2 Black Specks and Black Streaks

12.2.1 What Are Black Specks and Black Streaks?

Black specks and *black streaks* are dark spots or dark streaks found on the surface or throughout a molded part. Brown specks or streaks refer to the same type of defect, except the burning or discoloration is not as severe.

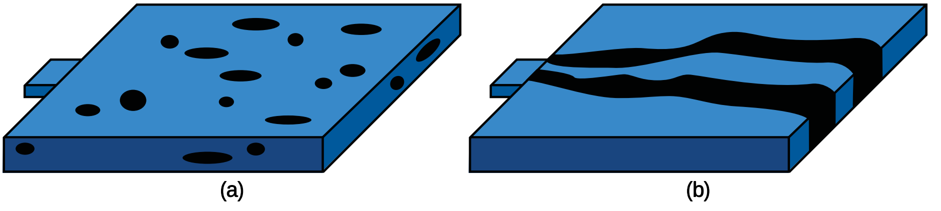


Figure 12.2 (a) Black specks and (b) Black streaks

12.2.2 Causes of Black Specks and Black Streaks

Black specks and black streaks are caused by overheated (degraded, burned) material or by contamination of the resin.

12.2.2.1 Material Degradation

Overheated materials can degrade and lead to black streaks. Material that stays in the nicked rough surfaces of the barrel wall and screw surfaces for a prolonged period of time after heating will char and degrade, resulting in the defect.

12.2.2.2 Material Contamination

Contaminants in the air or material, such as dirty regrind, foreign material, different color material, or a lower melt-temperature material, are what most often lead to black specks and black streaks. Airborne dirt can also cause dark spots on the surface of a molded part.

12.2.2.3 Other Defects Resulting from the Same Causes

- Brittleness
- Burn marks
- Discoloration

12.2.3 Remedies

12.2.3.1 Handle the Material Carefully

Make certain no contaminated materials, such as dirty regrind, are blended into the original material. Put the cover on the hopper and all bins of material. Airborne dirt can contaminate the original material, leading to black spots.

12.2.3.2 Alter the Mold Design

Clean the ejectors and slides. The streaks could by caused by the grease or lubricants on the slides or ejectors.

Improve the venting system. If the black specks are found at the end of flow paths or blind spots, they are likely caused by a poor venting system. Compressed, air trapped in the cavity is sometimes ignited, leading to the defect.

Clean or polish any nicked surface on the runner system to keep dirt from lodging in these areas.

Clean the mold before molding.

12.2.3.3 Select a Proper Machine

Size a proper injection machine for a specific mold. The typical shot size should be between 20 and 80% of machine injection capacity. For temperature-sensitive materials, the range should be narrowed down more. Plastics simulation software can help you select the right size injection machine for a specific mold. This will help avoid resin remaining in the heated barrel for prolonged periods of time.

Check for scratched or dented barrel/screw surfaces that trap material. This could lead to the material becoming overheated or burned.

Check for local overheating by a run-away heater band or a malfunctioning temperature controller.

12.2.3.4 Adjust the Molding Conditions

Lower the barrel and nozzle temperature. Material degradation can result from a high melt temperature.

Purge and clean the injection unit. The black streaks might be caused by contamination from the barrel wall or the screw surface. When molding with two materials, after switching from one material to the other, the old material might not be purged from the barrel completely. This could generate defects during the molding of the second material.

Avoid recycling rejected parts with black specks and black streaks. Recycling such parts could lead to further contamination, unless they will be used for parts that are in black or for which such defects are acceptable.

12.3 Brittleness

12.3.1 What Is Brittleness?

A brittle molded part has a tendency to break or crack. *Brittleness* results from material degradation leading to shorter molecular chain length (thus lower molecular weight). As a result, the physical integrity of the part is substantially less than the specification.

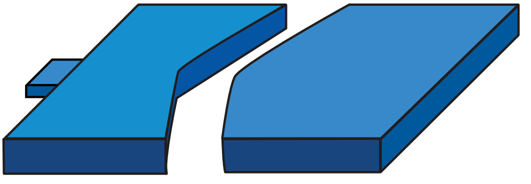


Figure 12.3 Degraded part tends to be brittle and break easily

12.3.2 Causes of Brittleness

Brittleness is caused by material degradation because of:

- Improper drying conditions
- Improper temperature setting
- Improper runner system and gate design
- Improper screw design
- Weld line weakness

12.3.2.1 Other Defects Resulting from the Same Causes

- Black specks and black streaks
- Burn marks
- Discoloration

12.3.3 Remedies

12.3.3.1 Adjust the Material Preparation

Set proper drying conditions before molding. Brittleness can be caused by excessive drying time or drying temperature such as at full heat for several days. Excessive drying either drives off volatiles in the plastic, making it more sensitive to processing, or degrades the material by reducing the molecular weight. Material suppliers can provide optimum drying conditions for the specific materials.

Reduce regrind material. The brittleness could be caused by too much reground material added into the original virgin material.

Change to a high-strength material since low-strength materials tend to become more brittle if processed improperly.

12.3.3.2 Alter the Mold Design

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

12.3.3.3 Select a Proper Machine/Machine Component

Get a better screw design for the material being used to achieve a better mixed melt temperature. Contact material suppliers to get the right screw design information to avoid improper melt mix or overheating that leads to material degradation.

12.3.3.4 Adjust Molding Conditions

Reduce the barrel temperature and nozzle temperature. If the barrel and nozzle temperature are too high, the material in the barrel will be overheated, leading to thermal degradation and the color change.

Reduce the back pressure, screw rotation speed, or injection speed because shear heating can result in material degradation.

While not overheating the material, increase melt temperature, mold temperature or injection pressure if the weld line has a tendency to crack. See Section 12.17 for more information about weld lines and meld lines.

12.4 Burn Marks

12.4.1 What Is a Burn Mark?

Burn marks are small, dark, or black spots that appear near the end of the molded part's flow path or in the blind area where an air trap forms.

Figure 12.4 Burn marks

12.4.2 Causes of Burn Marks

12.4.2.1 Entrapped Air

If the injection speed or injection pressure is too high, the air trapped in the runner system and cavity cannot be released to the atmosphere through the venting system properly within a very short filling time. Air traps also occur in improperly vented systems when race-tracking behavior is significant. Consequently, the air will be compressed, resulting in a very high pressure and temperature, which will cause the polymer to degrade on the surface near the end of the flow path or the blind area.

12.4.2.2 Material Degradation

Burn marks can also result from the degraded (charred) materials being carried downstream and then appearing on the surface of the molded part or near the venting areas. Material degradation is caused by:

- High melt temperature: Excessive melt temperature can be caused by improper barrel temperature setting, a broken thermocouple, or a malfunctioning temperature controller.
- High screw-rotation speed: If the screw speed is too high during the plasticization period, it will create too much frictional heat, which could degrade the material.
- Restrictive flow path: When the melt flows through restrictive nozzle, runner, gate, or part sections it creates a lot of shear (frictional) heat, which could degrade the material.

12.4.2.3 Other Defects Resulting from the Same Causes

- Black specks and black streaks
- Brittleness
- Discoloration

12.4.3 Remedies

12.4.3.1 Alter the Mold Design

Place an adequate venting system throughout the mold to help the entrapped air escape. Vents are especially important near the end of the flow path and in the blind area. The recommended venting size is 0.025 mm (0.001 in) for crystalline polymers and 0.038 mm (0.0015 in) for amorphous polymers.

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

12.4.3.2 Adjust the Molding Conditions

Reduce the likelihood of burn marks by avoiding excessive melt temperatures during the molding process:

- Reduce the injection pressure
- Reduce the injection speed
- Reduce the screw rotation speed
- Decrease the barrel temperature
- Check the band heaters on the barrel and nozzle, and calibrate the thermocouple

12.5 Delamination

12.5.1 What Is Delamination?

Delamination (sometimes called *lamination* or *layering*) is a defect in which the surface of a molded part can be peeled off layer by layer.

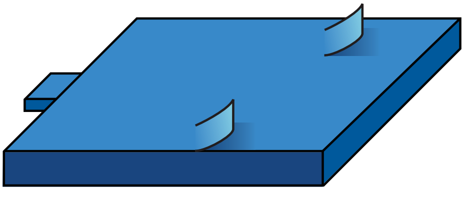


Figure 12.5 Delamination causes layer-wise peel-off on the surface of a molded part

12.5.2 Causes of Delamination

Delamination can be caused by several factors, including:

- Incompatible materials blended together
- Too much mold release agent being used during the molding process
- Low melt temperature in the cavity
- Excessive moisture
- Sharp corners at the gate and runner

12.5.3 Remedies

12.5.3.1 Change the Material Preparation

Avoid using foreign material or contaminated regrind material in the molding process.

12.5.3.2 Alter the Mold Design

Smooth all of the corners at the gate and runner. Sharp corners can tear apart melt flow and cause lamination.

12.5.3.3 Adjust the Molding Conditions

Avoid using excessive mold release agent to fix the de-molding problem. Delamination can be caused by excessive use of mold release agent. You should repair the ejection system or other problems to eliminate the difficulty of de-molding instead of overusing the mold release agent.

Follow the pre-dry instructions for the specific material and pre-dry the material properly before molding. Excessive moisture heats up and forms steam, which results in delamination on the surface.

Increase the barrel temperature and mold temperature. If the melt temperature is too low, layers of material are formed because they can't bond to each other. When ejected or subjected to stress, they separate from each other.

12.6 Dimensional Variation

12.6.1 What Is Dimensional Variation?

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

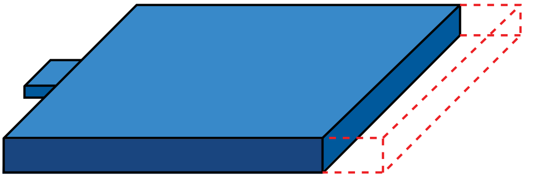


Figure 12.6 Dimensional variation is an unexpected change of part dimension

12.6.2 Causes of Dimensional Variation

Dimensional variation can be caused by:

- Unstable machine control
- A narrow molding window
- Improper molding conditions
- A broken check ring (within the injection unit)
- Unstable material properties

12.6.3 Remedies

12.6.3.1 Improve the Material Preparation

Contact the material vendor and change the material lot if the material has batch-to-batch variation.

Pre-dry the material before molding if the material is too wet.

Limit the percentage of regrind material added to the origin material. The irregular particle size can cause different levels of mixed melt material, and lead to unstable molded part dimensional variation.

12.6.3.2 Change a Mold Design or Component

Fix or adjust the ejection system if the molded part is bowed or distorted during ejection.

Design a proper runner and gate system for a specific mold and material. Use plastic injection molding simulation software to optimize the runner system dimensions to assure a smooth melt flow into the cavity.

12.6.3.3 Change a Machine Component

Replace the check ring if it is broken or worn out.

Replace heater bands or the thermocouple if it is out of order and causes unstable melt flow.

12.6.3.4 Adjust the Molding Conditions

Increase the injection and packing pressure. Make sure enough material is delivered into the cavity during the filling and packing stages.

Increase the injection and packing time to be sure enough material is delivered into the cavity during filling and packing stages.

Make sure the mold temperature is even by checking the cooling system.

Set up screw metering and injection stroke, screw-rotation speed, and back pressure properly so that they fall within the process window.

12.7 Discoloration

12.7.1 What Is Discoloration?

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

12.7.2 Causes of Discoloration

This defect can be caused by either material degradation or contamination from the following problems:

- The material stays in the barrel too long
- The barrel temperature is too high, causing the color to change
- Contamination was caused by reground material, different color material, or foreign material

12.7.2.1 Other Defects Resulting from the Same Causes

- Black specks and black streaks
- Brittleness
- Burn marks

12.7.3 Remedies

12.7.3.1 Handle the Material Carefully

Maintain proper housekeeping for virgin material and regrind material storage to avoid contamination of the materials.

12.7.3.2 Alter the Mold Design

Add an adequate venting system. To avoid discoloration (or burn marks) caused by poor venting or air traps, use the recommended venting size: 0.025 mm (0.001 in) for crystalline polymers and 0.038 mm (0.0015 in) for amorphous polymers.

12.7.3.3 Select a Proper Machine

Use a different size injection-molding machine. The typical shot size should be between 20 and 80% of machine injection capacity. For temperature-sensitive materials, the range should be narrowed down, depending on the material. Plastics simulation software can help you select the right size machine for a specific mold. This will help avoid the resin remaining in the heated barrel for prolonged periods of time.

12.7.3.4 Adjust the Molding Conditions

Clean the hopper completely. It is important to avoid foreign material or different color materials mixing together before molding.

Purge the injection unit completely if there is any material change.

Reduce the barrel and nozzle temperatures. If the barrel and nozzle temperatures are too high, the material in the barrel will be overheated, leading to thermal degradation and the color change.

12.8 Fish Eyes

12.8.1 What Are Fish Eyes?

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

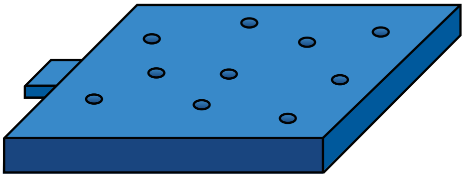


Figure 12.7 Unmelted materials in the melt stream causes fish eyes

12.8.2 Causes of Fish Eyes

Fish eyes are caused by:

- Low barrel temperature: If the barrel temperature is too low to melt the materials completely, the unmelted pellets will merge with the melt stream, marring the surface of the part.
- Too much regrind: The shape and size of regrind is irregular compared with original material, which can trap more air and cause the material to blend unevenly.
- Material contamination: If a high-melt-temperature material is blended into the original material, the blended material may stay in pellet form and cause fish eyes during the molding process.
- Low screw rotation speed and back pressure: If the screw rotation speed and the back pressure settings are too low, there might not be enough frictional heating to melt the material completely in the barrel before injection.

12.8.3 Remedies

12.8.3.1 Improve the Material Preparation

Limit or eliminate regrind for practical molding, depending on part quality requirements. Adding 10% of regrind is a good start, if regrind is allowed.

Store different materials separately and keep covers on the containers or bags to avoid blending different materials.

12.8.3.2 Adjust the Molding Conditions

Material suppliers usually provide the information about barrel temperature, back pressure, and screw rotation speed for specific materials. If you have followed suppliers' recommendations and are still experiencing problems, try making the following adjustments:

- Increase the barrel temperature
- Increase the back pressure to blend melt materials evenly
- Increase the screw rotation speed during the plasticization stage to create more frictional heat to melt materials

12.9 Flash

12.9.1 What Is Flash?

Flash is a defect characterized by excessive material found at locations where the mold separates, notably the parting surface, movable core, vents, or venting ejector pins.

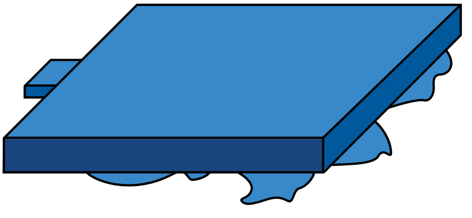


Figure 12.8 Flash

12.9.2 Causes of Flash

12.9.2.1 Low Clamp Force

If the clamp force of the injection machine is too low to hold the mold plates together during the molding process, flash will occur.

12.9.2.2 Gap within the Mold

Flash will occur if the parting surface does not contact completely, due to a deformed mold structure, parting surface defect, improper machine and mold set up, or flash or foreign material stuck on the parting surface.

12.9.2.3 Molding Conditions

Improper molding conditions, such as a high melt temperature (which makes a thinner melt) or high injection pressure, will cause flash.

12.9.2.4 Improper Venting

An improperly designed venting system, a very poor venting system, or a venting system that is too deep will cause flash.

12.9.3 Remedies

12.9.3.1 Adjust the Mold Set-up

Set up the mold to seal properly. A mismatch or undesirable gap between the cavity and core sides of the mold will result in flash.

Make sure the mold plates are strong enough to avoid deformation during molding. Add pillar support or thicken the mold plates if there is any deformation of the mold plate during the molding process.

Check for adequate venting dimensions.

The recommended venting size is 0.025 mm (0.001 in) for crystalline polymers and 0.038 mm (0.0015 in) for amorphous polymers.

Clean the mold surface. Flash can be caused by the mold surface not sealing well due to foreign material remaining between the parting surfaces.

Mill out the surface to keep the sealing pressure of land area around the cavities high enough.

12.9.3.2 Adjust the Machine Settings

Set up the machine and mold to seal properly. Flash can be caused by a poor seal between the cavity and core sides of the mold, and machine platens that are not parallel.

Increase the injection molding machine size. Flash can result from insufficient machine clamp force.

Adjust the clamp force if the machine capacity does have enough clamp force.

12.9.3.3 Adjust the Molding Conditions

Decrease the barrel temperature and nozzle temperature. A high melt temperature reduces the melt viscosity, making a thinner melt, which causes flash. But beware: avoid melt temperatures so low such that the resulting high injection pressure required causes flash.

Reduce the injection and packing pressure to reduce the clamp force requirement.

Reduce the feed setting (stroke length) to reduce metering (over-fill).

Increase the injection time or reduce the injection speed.

12.10 Flow Marks

12.10.1 What Is A Flow Mark?

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

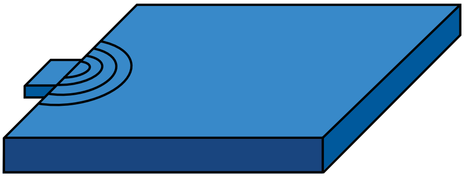


Figure 12.9 Flow mark

12.10.2 Causes of Flow Marks

Flow marks are caused by cold material near the gate or lack of compensated material during the packing stage. The problem usually can be attributed to:

- Low melt temperature
- Low mold temperature
- Low injection speed
- Low injection pressure
- Small runner stem and gate

According to a recent visual analysis using a glass-inserted mold, the flow mark defect can also be caused by cooling of the flow front portion on a cavity wall and the repeated phenomena of "getting over" and cooling with the subsequent melt. See Section 12.13 for more information about ripples.

12.10.3 Remedies

12.10.3.1 Alter the Mold Design

Change the size of the cold well in the runner system to trap the cold material during the filling stage. The proper length of the cold well is usually equal to that of the runner diameter.

Increase the runner system and gate size for the specific mold and material. Flow marks are sometimes caused by a restrictive runner system and gate size that freeze-off prematurely so that the material cannot be compensated during the packing stage.

Shorten the sprue length or use a hot runner design instead of a cold runner design.

12.10.3.2 Adjust the Molding Conditions

Increase the injection pressure and packing pressure.

Increase the barrel and nozzle temperature.

Increase the mold temperature.

12.11 Hesitation

12.11.1 What Is Hesitation?

Hesitation (or a *hesitation mark*) is a surface defect that results from the stagnation of polymer melt flow over a thin-sectioned area, or an area of abrupt thickness variation. Hesitation can be eliminated by changing the part thickness or moving the gate location.

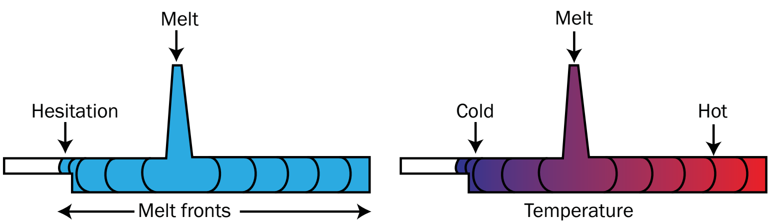


Figure 12.10 Hesitation results from stagnation of polymer melt flow

12.11.2 Problems Caused by Hesitation

When polymer melt is injected into a cavity of variable thickness, it tends to fill the thick and less resistant areas. As a result, polymer melt may stagnate at thin sections until the rest of the part is filled and the stagnated polymer melt starts moving again (see Figure 12.10). However, if the duration of hesitation is significant, polymer will solidify prematurely at the stagnated point. When the solidified melt front is pushed to the part surface, a surface defect such as a hesitation mark occurs.

12.11.3 Remedies

When troubleshooting the cause of hesitation in your part, both the part and mold design will need to be re-examined. Fine-tuning the processing conditions should also be tried.

12.11.3.1 Alter the Part Design

Reduce part thickness variation by changing wall thicknesses, if possible.

12.11.3.2 Alter the Mold Design

Position the gate away from the thin-sectioned areas, or regions of sudden thickness change. In this manner, hesitation will occur at a later time, and for a shorter duration. Figure 12.11 shows that in a poor design, hesitation results from stagnation of polymer melt flow; moving the gate away from the thin section reduces hesitation.

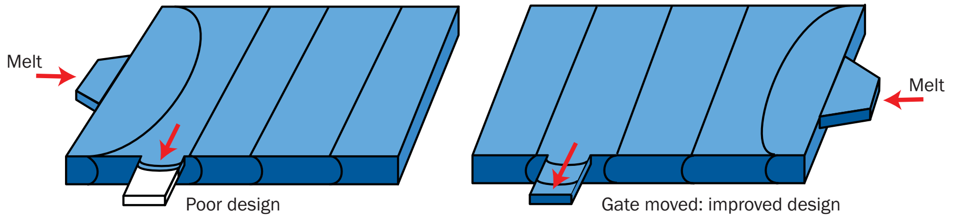


Figure 12.11 Hesitation resulting from improper gate position

12.11.3.3 Adjust the Molding Conditions

Increase the melt temperature and/or the injection pressure.

12.12 Jetting

12.12.1 What Is Jetting?

Jetting occurs when polymer melt is pushed at a high velocity through restrictive areas, such as the nozzle, runner, or gate, into open, thicker areas without forming contact with the mold wall. The buckled, snakelike jetting stream causes contact points to form between the folds of melt in the jet, creating small-scale "welds" (see Figure 12.12).

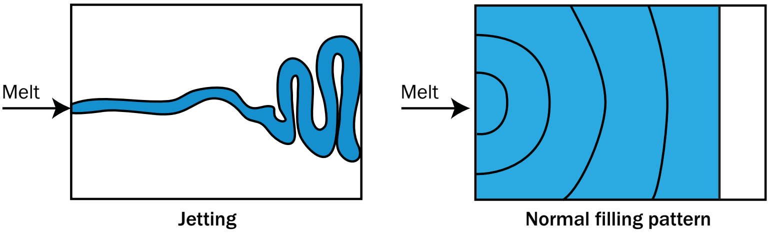


Figure 12.12 Jetting vs. normal filling pattern.

12.12.2 Problems Caused by Jetting

Jetting leads to part weakness, surface blemishes, and a host of internal defects. Contrast this with a normal filling pattern, in which melt advances in a progressive pattern from the gate to the extremities of the cavity, as illustrated in Figure 12.12.

12.12.3 Remedies

12.12.3.1 Alter the Mold Design

The trouble often lies with the gate design.

Direct the melt against a metal surface.

Use an overlap gate or a submarine gate as shown in Figure 12.13.

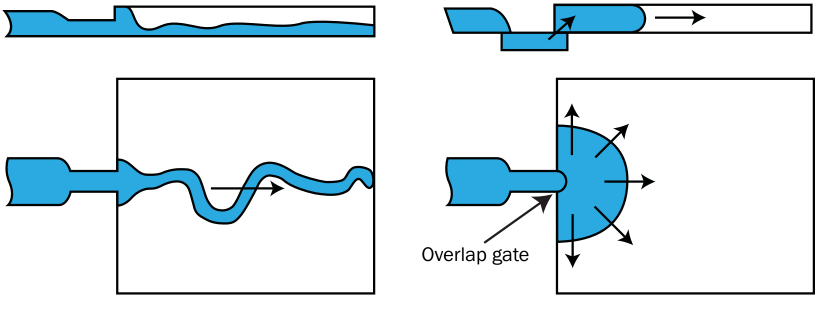


Figure 12.13 Using an overlap gate to avoid jetting

Slow down the melt with a gradually divergent flow area. A tab or fan gate provides a smooth transition from the gate to the cavity (Figure 12.14). This reduces the melt shear stress and shear rate.

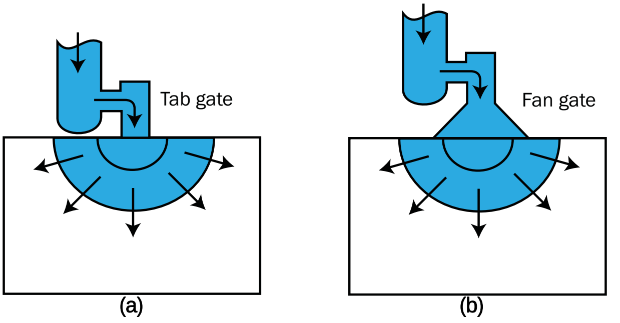


Figure 12.14 Using a tab gate (a) and a fan gate (b) to avoid jetting

Enlarge the size of the gate and runner or reduce the gate-land length. You can also relocate or redesign the gate in one of the following ways to reduce jetting.

12.12.3.2 Adjust the Molding Conditions

Adjust the ram-speed profile. Use an optimized ram-speed profile so that melt-front velocity is initially slow when the melt passes through the gate, then increases once a dispersed, tongue-shaped material is formed near the gate. Figure 12.15 illustrates this technique.

Adjust the barrel temperature to increase or decrease the melt temperature incrementally.

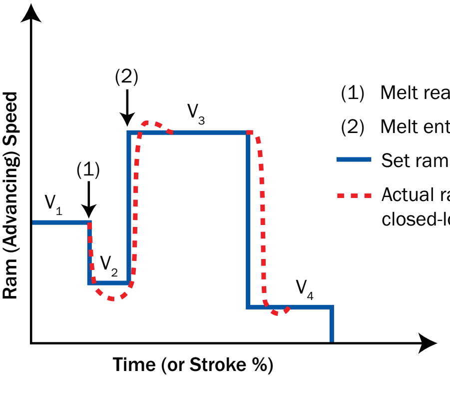


Figure 12.15 Adjust ram-speed profile to avoid jetting.

12.13 Ripples

12.13.1 What Are Ripples?

Ripples are the wavelets or small fingerprint-like waves near the edge or at the end of the flow.

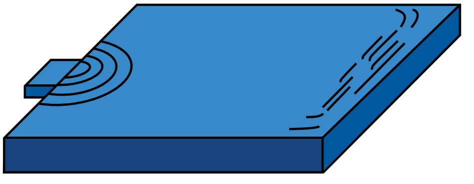


Figure 12.16 Ripples

12.13.2 Cause of Ripples

According to a recent visual analysis using a glass-inserted mold, the ripple defect is caused by the flow front portion of the melt cooling on a cavity wall, and the repeated phenomena of the subsequent melt "getting over" and cooling, as shown in Figure 12.17 below. Flow-front velocity and mold temperature have a stronger influence on the formation of ripples compared to the shape of the gates and the melt temperature.

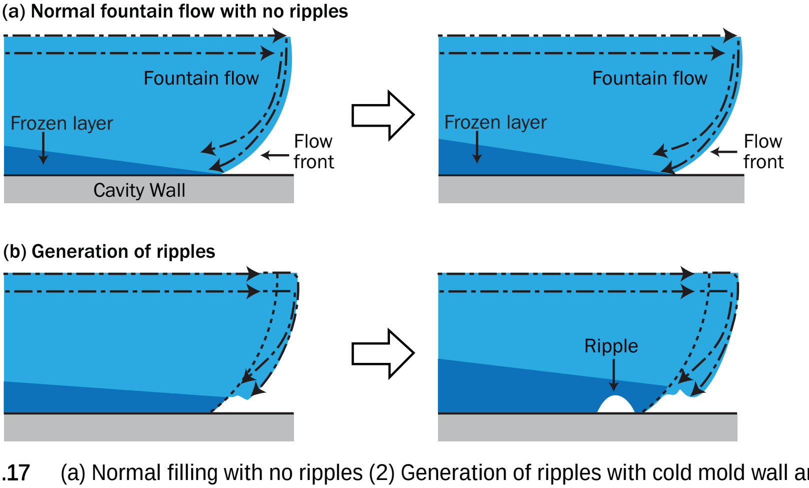


Figure 12.17 (a) Normal filling with no ripples (2) Generation of ripples with cold mold wall and low flow-front velocity

12.13.3 Remedies

Actions that increase the melt-front velocity or the mold and/or melt temperature will help to eliminate the ripples.

12.13.3.1 Modify the Part Design

Increase the part thickness.

12.13.3.2 Change the Mold Design

Make sure the runner system (including the sprue, runners, and gates) is adequate for the specific part.

Place an optimal venting system throughout the entire mold, especially around the end of the flow path. Make sure the venting system is large enough that the air present in the cavity can escape during injection. Be careful, however, that the venting system is not so large that it causes flash at the edge of the molding. The recommended venting size is 0.025 mm (0.001 in) for crystalline polymers, and 0.038 mm (0.0015 in) for amorphous polymers.

12.13.3.3 Adjust the Molding Conditions

Increase the mold temperature.

Increase the injection speed. This will create more viscous heating and reduce the melt viscosity.

Increase the injection pressure. Be careful not to exceed the machines's capacity. The operating injection pressure should normally be limited to 70 to 85% of the maximum injection pressure to prevent accidental damage to the machine's hydraulic system.

Increase the melt temperature. Be careful not to introduce material degradation from prolonged exposure at an elevated temperature.

12.14 Short Shots

12.14.1 What Is a Short Shot?

A *short shot* is a molded part that is incomplete because insufficient material was injected into the mold. In some cases, short shots are intentionally produced to determine or visualize the filling pattern. But problematic short shots occur when the polymer melt cannot fill the entire cavity (or cavities). This most commonly occurs at thin sections or extremities.

12.14.2 Causes of Short Shots

Any factors that increase the resistance of polymer melt to flow or prohibit delivery of sufficient material into the cavity can cause a short shot. These factors include:

- Insufficiently-sized restrictive-flow areas, such as gates, runners, and thin walls
- Low melt and/or mold-wall temperatures
- A lack of vents to bleed the air trapped inside the cavity
- Insufficient machine injection pressure (resulting from high melt resistance and a restricted flow path), volume, and/or ram speed
- Machine defects such as an empty hopper, blocked feed throat, or a worn nonreturn (check) valve that causes loss of injection pressure or leakage of injection volume
- Premature solidification of the polymer melt due to hesitation, poor filling pattern, or prolonged injection time

12.14.3 Remedies

Several factors influence the polymer's ability to fill the entire cavity. Proper remedial actions can be taken when the cause of a short shot is pinpointed. Here are some suggestions.

12.14.3.1 Alter the Part Design

It is important to facilitate the flow of injected polymer melt; doing so can alleviate short shots. Strategically increase the thickness of certain wall sections (as flow leaders).

12.14.3.2 Alter the Mold Design

A properly designed delivery system (sprue, runner, and gate) will facilitate a more balanced filling pattern. If needed, modify your design in the following ways:

- Fill the thick areas before filling the thin areas. Doing so will avoid hesitation, which causes early solidification of polymer.
- Increase the number and/or size of gates to reduce the flow length.
- Increase the size of runner systems to reduce resistance.

Entrapped air inside the mold cavity (air traps) can also lead to short shots.

- Place vents at the proper locations, typically near the areas that fill last. This should help vent the displaced air.
- Increase the size and number of vents.

12.14.3.3 Adjust the Molding Conditions

Look closely at the factors that control how material is injected into the mold.

- Increase the injection pressure. Do not exceed the machine's capability. To prevent accidental damage to the machine's hydraulic system, limit the operating injection pressure to 70 to 85% of the maximum injection pressure.
- Increase the injection speed. Within the machine limits, this will create more viscous heating and reduce the melt viscosity.
- Increase the injection volume.
- Increase the barrel temperature and/or the mold-wall temperature. Higher temperatures will promote the flow of material through the cavity. Be careful to avoid material degradation due to prolonged exposure at an elevated temperature.

The molding machine might also be the culprit if you are experiencing problematic short shots.

- Check the hopper for sufficient material supply or a clogged feed throat.
- Inspect the non-return valve and barrel for excessive wear. Wear can lead to loss of injection pressure and leakage of injection volume.

12.15 Silver Streaks

12.15.1 What Are Silver Streaks?

Silver streaks are the splash appearance of moisture, air, or charred plastic particles on the surface of a molded part, which are fanned out in a direction emanating from the gate location.

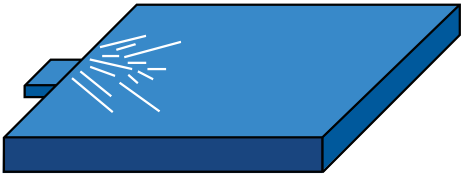


Figure 12.18 Silver streaks

12.15.2 Causes of Silver Streaks

Silver streaks can be caused by:

- Moisture: Plastic materials absorb a certain degree of moisture during storage. If the material is not dried properly before molding, the moisture residing in the resin will turn into a steam during the injection process and splay on the surface of the molded part.
- Air: During the plasticization period, a certain amount of gas can be trapped and blended into the melt material. If the air does not escape during the injection process, it could splay out on the surface of the molded part.
- Degraded (charred) plastic particles: There are a several reasons degraded (charred) plastic particles will splay on the surface of a molded part:
  - ➤ Material contamination: When molding with two materials, as you switch from one material to another, the residual particles left in the barrel could be charred if the second material is being molded at a higher temperature. In addition, contaminated, rejected parts and regrind will re-contaminate virgin material in the next batch of molded parts.
  - ➤ Barrel temperature: An improper barrel temperature setting may degrade polymer molecules, and they will begin to char.
  - ➤ Shot volume: If the shot size is below 20% of the machine injection capacity, especially for temperature-sensitive materials, the melt resin will remain in the barrel too long and will begin to degrade.

12.15.3 Remedies

12.15.3.1 Handle the Material Carefully

Dry the material properly before molding, according to the resin supplier's instructions.

12.15.3.2 Alter the Mold Design

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

Check for adequate venting dimensions. The recommended venting size is 0.025 mm (0.001 in) for crystalline polymers, and 0.038 mm (0.0015 in) for amorphous polymers.

12.15.3.3 Adjust the Molding Conditions

The following precautions will deter material from degrading during the process:

- Size a proper injection machine for a specific mold. The typical shot size should be between 20 and 80% of the machine injection capacity. For temperature-sensitive materials, the range should be narrowed down, depending on materials. Plastics simulation software can help you select the right size injection machine for a specific mold. This will help to avoid a prolonged residence time for resin in the heated barrel.

- Fully purge the older material from the barrel if switching material from one to the other. Old material particles left behind could be charred.
- Increase the back pressure. This will help minimize air blending into the melt material.
- Improve the venting system. It is important to allow air and steam to escape easily.
- Decrease the melt temperature, injection pressure, or injection speed.

12.16 Sink Marks and Voids

12.16.1 What Are Sink Marks and Voids?

A *sink mark* is a local surface depression that typically occurs in moldings with thicker sections, or at locations above ribs, bosses, and internal fillets. A *void* is a vacuum bubble in the core.

12.16.2 Causes of Sink Marks and Voids

Sink marks and voids are caused by localized shrinkage of the material at thick sections without sufficient compensation when the part is cooling. A sink mark almost always occurs on a surface that is opposite to and adjoining a leg or rib. This occurs because of unbalanced heat removal or similar factors.

Factors that lead to sink marks and voids are:

- Low injection and packing pressure
- Short hold time or cooling time
- High melt temperature or mold temperature
- Localized geometric features

After the material on the outside has cooled and solidified, the core material starts to cool. Its shrinkage pulls the surface of the main wall inward, causing a sink mark. If the skin is rigid enough, as in engineering resins, deformation of the skin may be replaced by formation of a void in the core. Figure 12.19 illustrates this phenomenon.

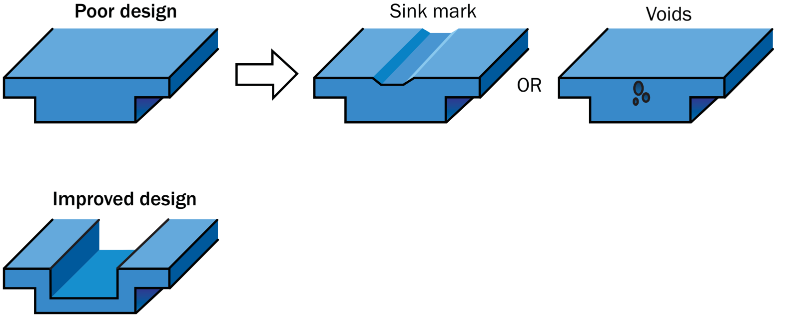


Figure 12.19 Sink marks and voids are created by material shrinkage without sufficient compensation

12.16.3 Remedies

Sink marks and voids can usually be alleviated by fine-tuning some combination of your part and mold design and the conditions under which the part is molded. Use the suggestions below to pinpoint and fix the problem.

12.16.3.1 Alter the Part Design

Conceal sink marks by adding a design feature, such as a series of serrations on the area where they occur. Figure 12.20 illustrates this technique.

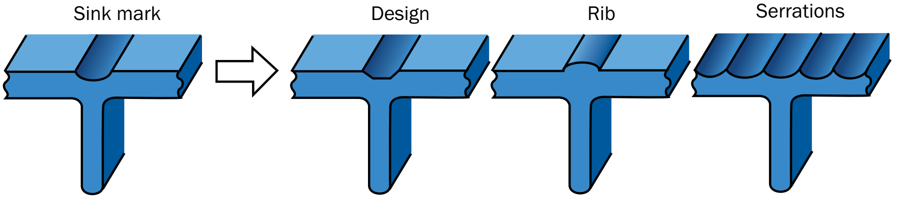


Figure 12.20 Sink marks can be eliminated by creating a design feature, rib, or serrations

Modify the part thickness design as suggested to minimize the thickness variation.

Re-design the thickness of the ribs, bosses, and gussets to be 50 to 80% of the attached (base) wall thickness. Figure 12.21 shows the dimensions we prescribe.

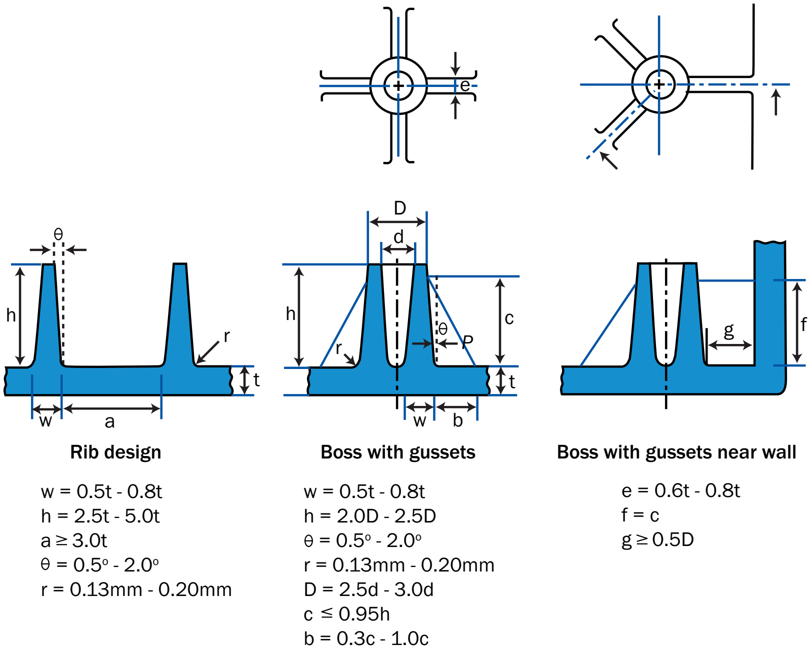


Figure 12.21 Recommended dimensions for ribs, bosses, and gussets

12.16.3.2 Alter the Mold Design

Increase the size of gates and runners to delay the gate freeze-off time. This allows more material to be packed into the cavity.

Add more vents or enlarge the existing vents. Vents allow air trapped inside the cavity to escape.

Relocate the gate to or near a thicker section. This allows them to be packed before the thinner sections freeze off.

12.16.3.3 Adjust the Molding Conditions

Increase the cushion at the end of the injection stroke. You should maintain a cushion of approximately 3 mm (0.12 in).

Increase the injection pressure and the holding time.

Increase the screw-forward time and decrease the injection rate.

Decrease the melt and mold-wall temperatures.

Increase the cooling time.

Check the non-return valve for possible material leakage.

12.17 Weld Lines and Meld Lines

12.17.1 What Are Weld Lines and Meld Lines?

A *weld line* (also called a *weld mark* or a *knit line*) is formed when separate melt fronts traveling in opposite directions meet. A *meld line* occurs if two emerging melt fronts flow parallel to each other and create a bond between them. Weld and meld lines can be caused by holes or inserts in the part, multiple gates, or variable wall thickness where hesitation or race tracking occurs. If weld or meld lines cannot be avoided, position them at low-stress and low-visibility areas by adjusting the gate position. Improve the strength of weld and meld lines by increasing the local temperature and pressure at their locations.

12.17.1.1 How to Tell the Difference between Weld Lines and Meld Lines

Traditionally, the *meeting angle* is used to differentiate weld lines and meld lines. As illustrated in Figure 12.22, a meeting angle, θ, smaller than 135° produces a weld line; greater than 135°, a meld line. Normally, weld lines are considered to be of lower quality than meld lines, since relatively less molecular diffusion occurs across a weld line after it is formed.

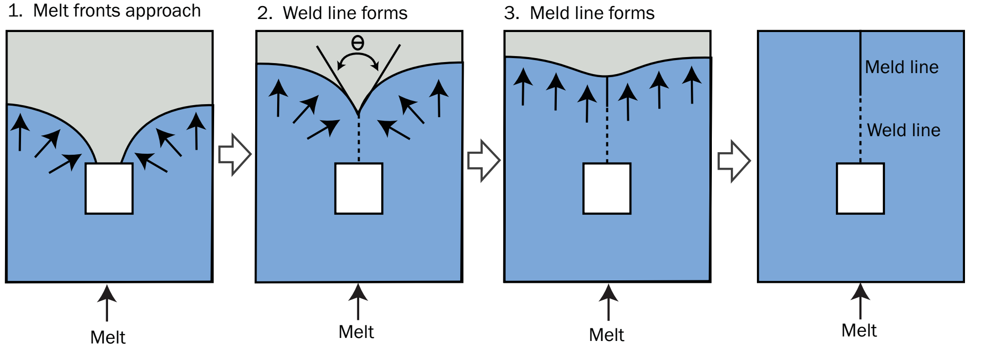


Figure 12.22 Weld and meld lines

12.17.2 Problems Caused by Weld Lines

Weld lines are generally undesirable when part strength and surface appearance are major concerns. This is especially true with fiber-reinforced materials, because the fibers do not bridge the weld lines and often are oriented parallel to them, as illustrated in Figure 12.23.

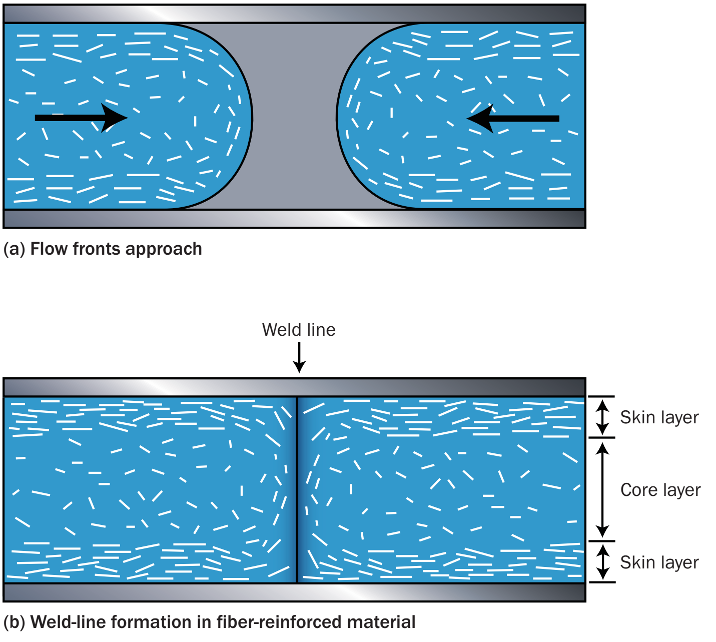


Figure 12.23 Fiber distribution parallel to the weld line leads to a weaker bond

12.17.3 Strength of Weld Lines

The exact strength of the weld line depends on the ability of the flow fronts to weld (or knit) to each other. The strength of the weld-line area can be from 10 to 90% as strong as the pure material used. With such a wide range possible, the conditions that are favorable to better weld line quality are worth examining:

- High injection pressure and speed
- High melt and mold-wall temperature
- Formation of the weld lines closer to the gate
- A temperature difference of less that 10°C between the two emerging flow fronts.

If a weld line forms before the filling is complete and is immediately subject to additional packing pressure, the weld line will typically be less visible and stronger. For complex part geometry, flow simulation helps to predict the weld/meld line position with respect to changes in the tool design and to monitor the temperature difference.

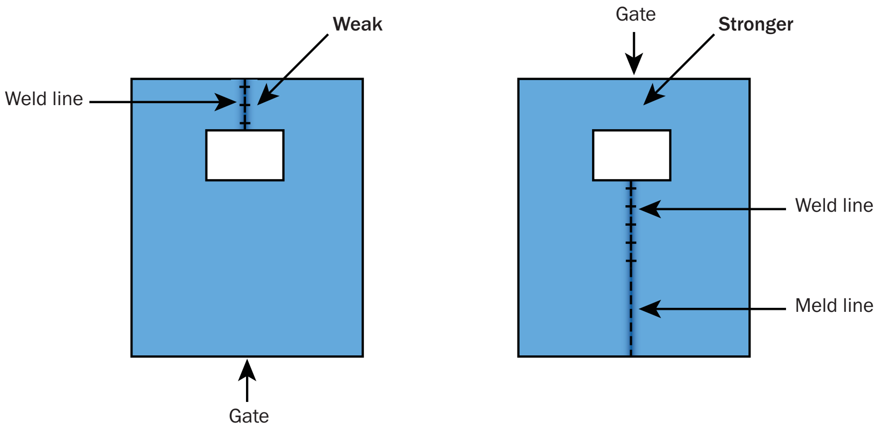


Figure 12.24 Changing the weld-line position by modifying the gate location

12.17.4 Remedies

12.17.4.1 Alter the Part Design

Increase the wall thickness. This will facilitate the transmission of pressure and maintain a higher melt temperature.

Adjust the gate position.

12.17.4.2 Alter the Mold Design

Increase the size of gate and runners.

Place a vent in the area of the weld/meld line. This will eliminate entrapped air, which would further weaken the weld/meld-line.

Change the gate design to eliminate weld/meld lines or to form them closer to the gate at a higher temperature and under a higher packing pressure.

12.17.4.3 Adjust the Molding Conditions

Increase the melt temperature, injection speed, or injection pressure.

Appendix A: Injection Molding

- Injection-molding overview
- Development of the injection-molding machine
- Development of the injection-molding process
- Co-injection (sandwich) molding
- Fusible core injection molding
- Gas-assisted injection molding
- Injection-compression molding
- Lamellar (microlayer) injection molding
- Live-feed injection molding
- Low-pressure injection molding
- Push-pull injection molding
- Reactive molding
- Structural foam injection molding
- Thin-wall molding

A.1 Injection-molding Overview

A.1.1 Process

Injection molding is a cyclic process of forming plastic into a desired shape by forcing material under pressure into a cavity. The shaping is achieved by cooling (thermoplastics) or a chemical reaction (thermosets). It is one of the most common and versatile operations for mass producing complex plastics parts with excellent dimensional tolerance. It requires minimal or no finishing or assembly operations. In addition to thermoplastics and thermosets, such materials as fibers, ceramics, and powdered metals are also being used with polymers as binders.

A.1.2 Applications

By weight, approximately 32% of all plastics processed go through injection-molding machines. Historically, the invention of various new alternative processes, such as the reciprocating screw machine, and the application of computer simulation to the design and manufacture of plastics parts are the major milestones of injection molding.

A.2 Development of the Injection-molding Machine

Since its introduction in the early 1870s, the injection-molding machine has undergone significant modifications and improvements. In particular, the invention of the reciprocating screw machine has revolutionized the versatility and productivity of the thermoplastic injection-molding process.

A.2.1 Benefits of the Reciprocating Screw

Apart from obvious improvements in machine control and machine functions, the major development for the injection-molding machine is the change from a plunger mechanism to a reciprocating screw. Although the plunger-type machine is inherently simple, its popularity was limited because of the slow heating rate through pure conduction only. The reciprocating screw, as shown in Figure A.1, can plasticize the material more quickly and uniformly with its rotating motion In addition, it is able to inject the molten polymer in a forward direction, as a plunger.

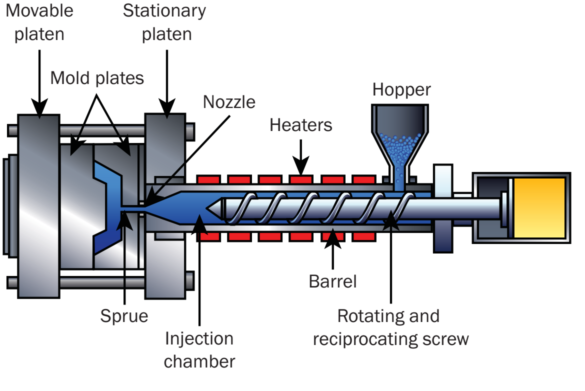


Figure A.1 The reciprocating-screw injection-molding machine

A.3 Development of the Injection-molding Process

The injection-molding process was first used only with thermoplastic polymers. Advances in the understanding of materials, improvements in molding equipment, and the needs of specific industry segments have expanded the use of the process to areas beyond its original scope.

A.4 Alternative Injection-molding Processes

During the past several decades, numerous attempts have been made to develop injection-molding processes to produce parts with special design features and properties. Alternative processes derived from conventional injection molding have created a new era for additional applications, more design freedom, and special structural features. These efforts have resulted in a number of processes, including:

- Co-injection (sandwich) molding (Section A.4.1)
- Fusible core injection molding (Section A.4.2)
- Gas-assisted injection molding (Section A.4.3)
- Injection-compression molding (Section A.4.4)
- Lamellar (microlayer) injection molding (Section A.4.5)
- Live-feed injection molding (Section A.4.6)
- Low-pressure injection molding (Section A.4.7)
- Push-pull injection molding (Section A.4.8)
- Reactive molding (Section A.4.9)
- Structural foam injection molding (Section A.4.10)
- Thin-wall molding (Section A.4.11)

A.4.1 Co-injection (Sandwich) Molding

Co-injection molding involves sequential or concurrent injection of two different but compatible polymer melts into a cavity. The materials laminate and solidify. This process produces parts that have a laminated structure, with the core material embedded between the layers of the skin material. This innovative process offers the inherent flexibility of using the optimal properties of each material or modifying the properties of the molded part.

Figure A.2 shows the steps involved in the co-injection molding process.

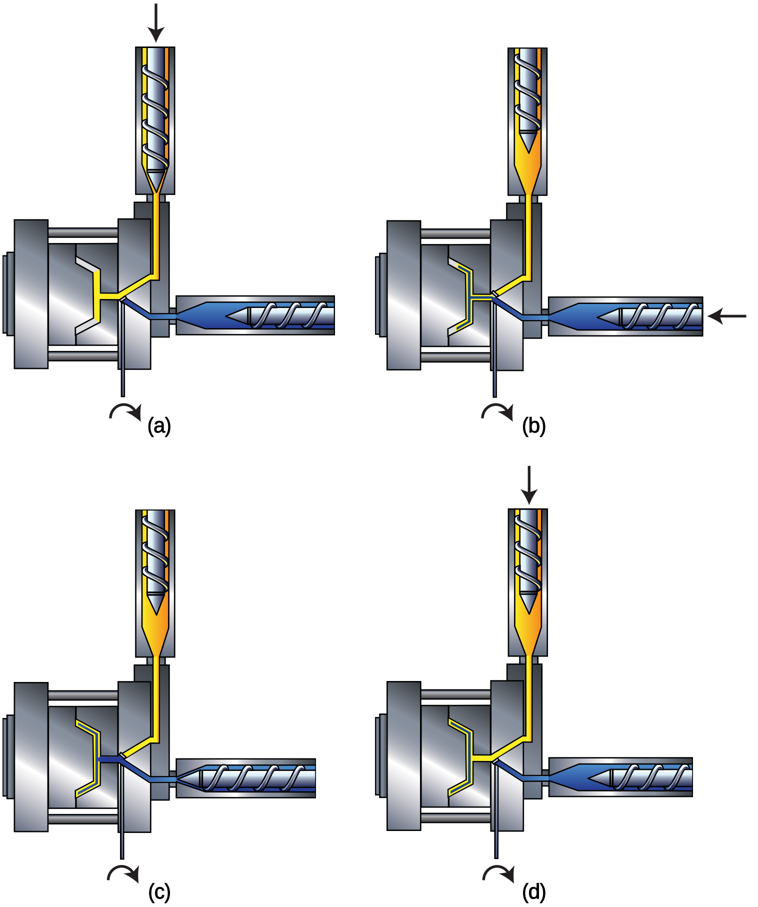


Figure A.2 Four stages of co-injection molding: (a) Short shot of skin polymer melt is injected into the mold. (b) Injection of core polymer melt until cavity is nearly filled, as shown in (c). (d) Skin polymer is injected again to purge the core polymer away from the sprue.

A.4.2 Fusible Core Injection Molding

The fusible (lost, soluble) core injection-molding process illustrated below produces single-piece, hollow parts with complex internal geometry. This process molds a core inside the plastic part. After the molding, the core will be physically melted or chemically dissolved, leaving its outer geometry as the internal shape of the plastic part.

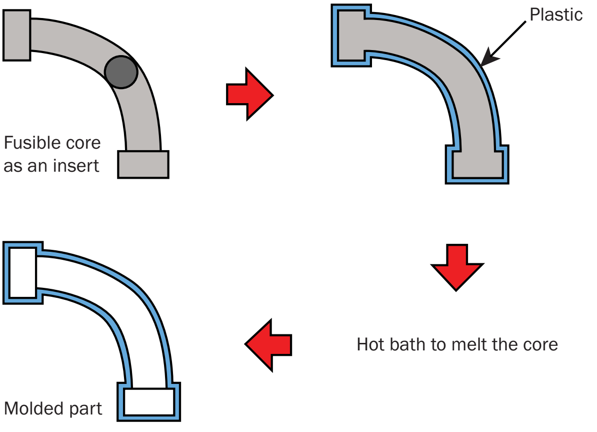


Figure A.3 Fusible (lost, soluble) core injection molding

A.4.3 Gas-assisted Injection Molding

The gas-assisted injection molding process begins with a partial or full injection of polymer melt into the mold cavity. Compressed gas is then injected into the core of the polymer melt to help fill and pack the mold. This process is illustrated in Figure A.4.

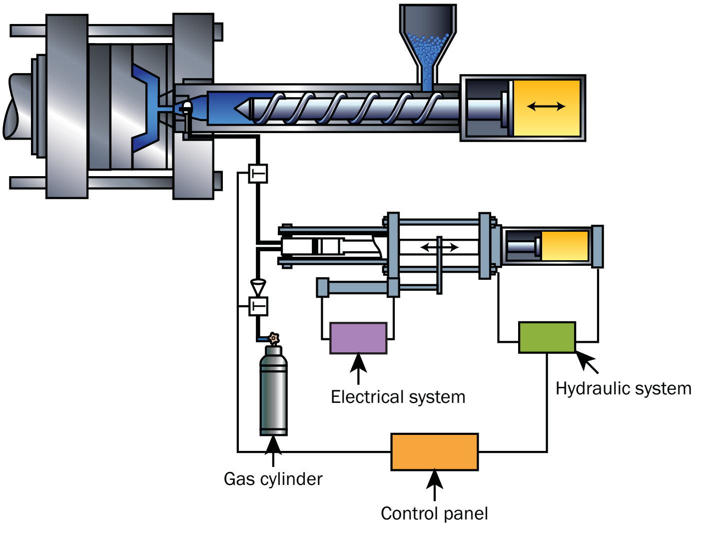


Figure A.4 Gas-assisted injection molding

A.4.3.1 Benefits of the Gas-assist Process

The gas-assisted injection-molding process is capable of producing hollow, light-weight, rigid parts that are free of sink marks and less likely to warp. Other advantages include:

- Reduced cycle time
- Reduced pressure and clamp force tonnage
- Part consolidation with both thick and thin sections

A.4.3.2 Typical Applications

Typical applications for the gas-assisted injection-molding process can be classified into three categories, or some combination of them:

- Tube-and rod-like parts, where the process is used primarily for saving material, reducing the cycle time by coring out the part, and incorporating the hollowed section with product function. Examples are clothes hangers, grab handles, chair armrests, shower heads, and water faucet spouts.
- Large, sheet-like, structural parts with a built-in gas-channel network, where the process is used primarily for reducing part warpage and clamp tonnage as well as to enhance rigidity and surface quality. Examples are automotive panels, business machine housings, outdoor furniture, and satellite dishes.
- Complex parts consisting of both thin and thick sections, where the process is used primarily for decreasing manufacturing cost by consolidating several assembled parts into one single design. Examples are television cabinets, computer printer housing bezels, and automotive parts.

A.4.4 Injection-compression Molding

The injection-compression molding process is an extension of conventional injection molding. After a pre-set amount of polymer melt is fed into an open cavity, it is compressed, as shown in Figure A.5. The compression can also take place when the polymer is to be injected. The primary advantage of this process is the ability to produce dimensionally stable, relatively stress-free parts at a low clamp tonnage (typically 20 to 50% lower).

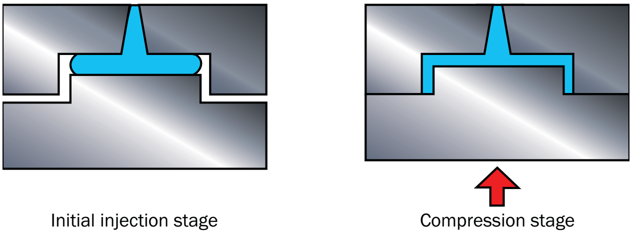


Figure A.5 Injection-compression molding

A.4.5 Lamellar (Microlayer) Injection Molding

This process uses a feedblock and layer multipliers to combine melt streams from dual injection cylinders. It produces parts from multiple resins in distinct microlayers, as shown in Figure A.6. Combining different resins in a layered structure enhances a number of properties, such as the gas barrier property, dimensional stability, heat resistance, and optical clarity.

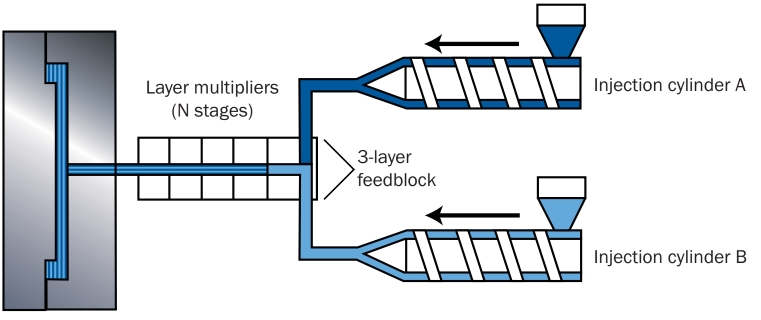


Figure A.6 Lamellar (microlayer) injection molding

A.4.6 Live-feed Injection Molding

The live-feed injection molding process applies oscillating pressure at multiple polymer entrances to cause the melt to oscillate, as shown in Figure A.7. The action of the pistons keeps the material in the gates molten while different layers of molecular or fiber orientation are being built up in the mold from solidification. This process provides a means of making simple or complex parts free from voids, cracks, sink marks, and weld-line defects.

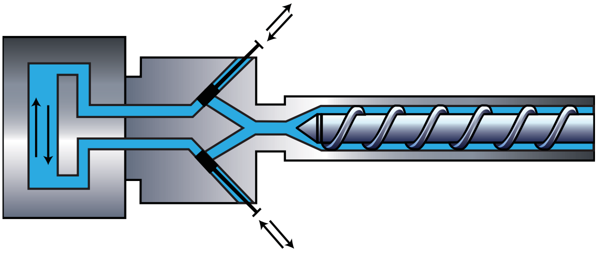


Figure A.7 Live-feed injection molding

A.4.7 Low-pressure Injection Molding

Low-pressure injection molding is essentially an optimized extension of conventional injection molding (see Figure A.8). Low pressure can be achieved by properly programming the screw revolutions per minute, hydraulic back pressure, and screw speed to control the melt temperature and the injection speed. It also makes use of a generous gate size or a number of valve gates that open and close sequentially to reduce the flow length. The packing stage is eliminated with a generally slow and controlled injection speed. The benefits of low-pressure injection molding include a reduction of the clamp force tonnage requirement, less costly molds and presses, and lower stress in the molded parts.

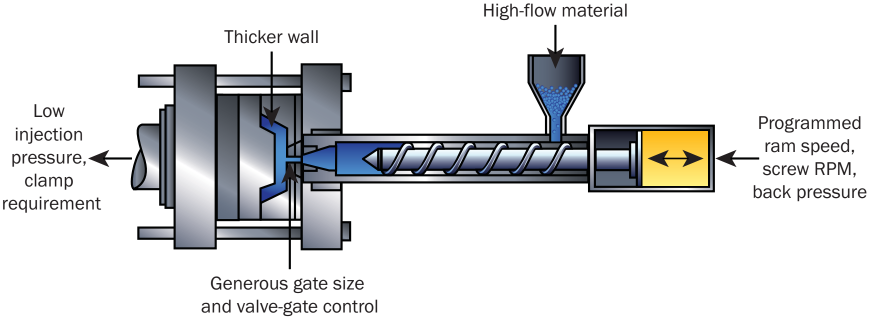


Figure A.8 Low-pressure injection molding

A.4.8 Push-pull Injection Molding

The push-pull injection molding process uses a conventional twin-component injection system and a two-gate mold to force material to flow back and forth between a master injection unit and a secondary injection unit, as shown in Figure A.9. This process eliminates weld lines, voids, and cracks, and controls the fiber orientation.

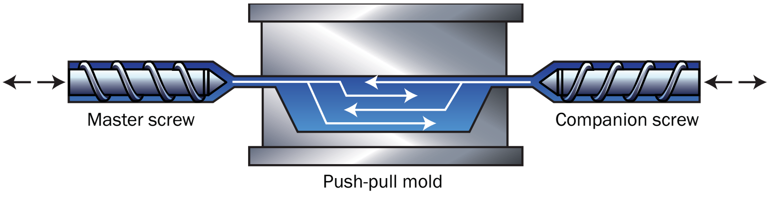


Figure A.9 Push-pull injection molding

A.4.9 Reactive Molding

Unlike thermoplastics, reactive materials undergo simultaneous forming and polymerization during the molding process. The cross-linked polymer structure generally imparts improved mechanical properties and greater heat and environmental resistance.

Table A.1: Types of reactive materials

Acrylic	Phenolics
Alkyds	Polyurethane polyols
Allyl diglycol carbonate	Polyurethane isocyanates
DAIP	Polyurethane systems
DAP	Silicone
Epoxy	Silicone/polymide
Fluorosiliconeurea	Urea
Melamine	Unsaturated polyesters
Melamine/phenolic	

A.4.9.1 Processing

Major reactive molding processes include reactive injection molding (RIM) and composites processing, such as resin transfer molding (RTM) and structural reactive injection molding (SRIM). The typically low viscosity of the reactive materials permits large and complex parts to be molded with relatively lower pressure and clamp tonnage than required for thermoplastics molding. Reactive resins can also be used in the composite processes. For example, to make high-strength and low-volume large parts, RTM and SRIM can be used to include a preform made of long fibers. The encapsulation of microelectronic IC chips is another area receiving more attention than ever before.

The adaptation of injection molding to these materials includes only a small increase in the feed mechanism (barrel) temperature to avoid pre-curing. The cavity, however, is usually hot enough to initiate chemical cross-linking. As the warm pre-polymer is forced into the cavity, heat is added from the cavity wall from both the viscous (frictional) heating of the flow and the heat released by the reacting components. The temperature of the part often exceeds the temperature of the mold. When the reaction is sufficiently advanced for the part to be rigid (even at a high temperature) the cycle is complete and the part is ejected.

A.4.9.2 Design Considerations

The mold and process design for injection molding of reactive materials is much more complex because of the chemical reaction that takes place during the filling and packing stages. For instance, slow filling often causes premature gelling and a resultant short shot, while fast filling could induce the turbulent flow that creates internal porosity. Improper control of mold-wall temperature and/or inadequate part thickness will cause either moldability problems during injection or scorching of the materials. Computer simulation is generally recognized as a more cost-effective tool than the conventional and time-consuming trial-and-error method for tool and process debugging.

A.4.10 Structural Foam Injection Molding

Structural foam molding produces parts consisting of solid external skin surfaces surrounding an inner cellular (or foam) core, as illustrated in Figure A.10. This process is suitable for large, thick parts subject to bending loads in their end-use application. Structural foam parts can be produced with both low and high pressure, using nitrogen gas or chemical blowing agents.

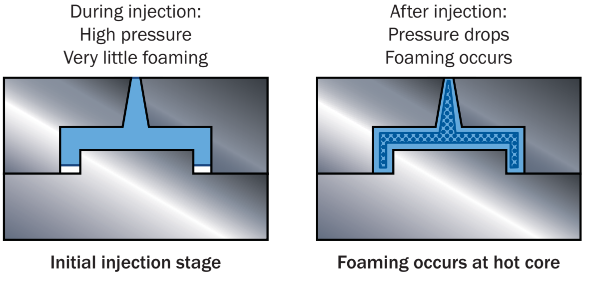


Figure A.10 Structural foam injection molding

A.4.11 Thin-wall Molding

The term *thin-wall* is relative. Conventional plastic parts are typically 2 to 4 mm thick. Thin-wall designs are called *advanced* when thicknesses range from 1.2 to 2 mm and *leading-edge* when the dimension is below 1.2 mm. Another definition of thin-wall molding is based on the flow-length-to-wall-thickness ratios. Typical ratios for these thin-wall applications range from 100:1 to 150:1 or more.

A.4.11.1 Typical Applications

Thin-wall molding is popular in portable communication and computing equipment, which demand very thin plastic shells that still provide the same mechanical strength as conventional parts.

A.4.11.2 Processing

Because thin-wall parts freeze off quickly, they require high melt temperatures, high injection speeds, and very high injection pressures if multiple gates or sequential valve gating are not used. An optimized ram-speed profile helps to reduce the pressure requirement.

Due to the high velocity and shear rate in thin-wall molding, orientation occurs more readily. To help minimize anisotropic shrinkage in thin-wall parts, it is important to pack the part adequately while the core is still molten.

Molders should watch for excessive residence time, melt temperatures, or shear-all of which can cause material degradation.

A.4.11.3 Design

We recommend designing parts with styling lines and curved surfaces to boost stiffness and enhance part aesthetics. Impact strategies involve using unreinforced plastic housing to absorb the load or using filled thermoplastics to transfer it. For either case, you'll need to fasten internal components snugly and avoid stress concentration and sharp notches.

Large gates, greater than the wall thickness, are generally used to ensure sufficient material flow during packing.

A.4.11.4 Computer Simulation

To simulate thin-wall molding accurately under high pressure, high injection speed, and fast cooling conditions users should specify the following:

- Pressure dependence of viscosity, to account for melt viscosity increases with increasing pressure
- Spacial variation of the density (PVT or fast-cooling PVT), to account for the pressure and temperature dependence of density or the effect of fast cooling rate
- Compression work, to account for the additional heating due to the compression work

Appendix B: Injection-molding Machine: System and Operations

- Injection-molding machine
- Machine components
- Molded system
- Machine operating sequence
- Screw operation
- Secondary operations

B.1 Injection-molding Machine

B.1.1 Components

For thermoplastics, the injection-molding machine converts granular or pelleted raw plastic into final molded parts via a melt, inject, pack, and cool cycle. A typical injection molding machine consists of the following major components, as illustrated in Figure A.1.

- Injection system
- Hydraulic system
- Mold system
- Clamping system
- Control system

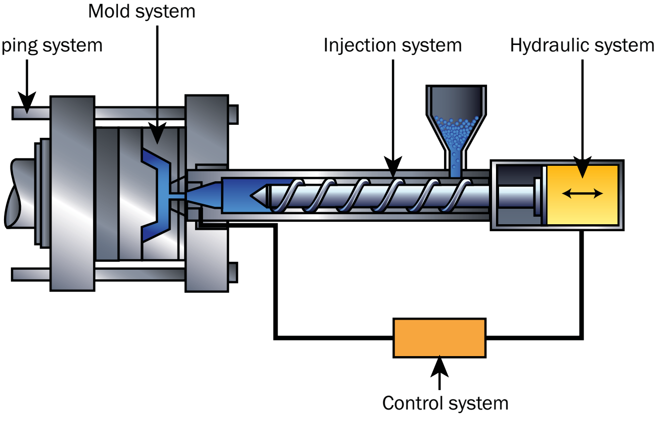


Figure A.1 A single screw injection-molding machine for thermoplastics

B.1.2 Machine Specification

Clamp tonnage and shot size are commonly used to quickly identify the size of the injection-molding machine for thermoplastics. Other parameters include injection rate, injection pressure, screw design, mold thickness, and the distance between tie bars.

B.1.3 Machine Function

Injection-molding machines can be generally classified into three categories, based on machine function:

- General-purpose machines
- Precision, tight-tolerance machines
- High-speed, thin-wall machines

B.1.4 Auxiliary Equipment

The major equipment auxiliary to an injection-molding machine includes resin dryers, material-handling equipment, granulators, mold-temperature controllers and chillers, part-removal robots, and part-handling equipment.

B.2 Machine Components

B.2.1 Injection System

The injection system consists of a hopper, a reciprocating screw and barrel assembly, and an injection nozzle, as shown in Figure A.2. This system confines and transports the plastic as it progresses through the feeding, compressing, degassing, melting, injection, and packing stages.

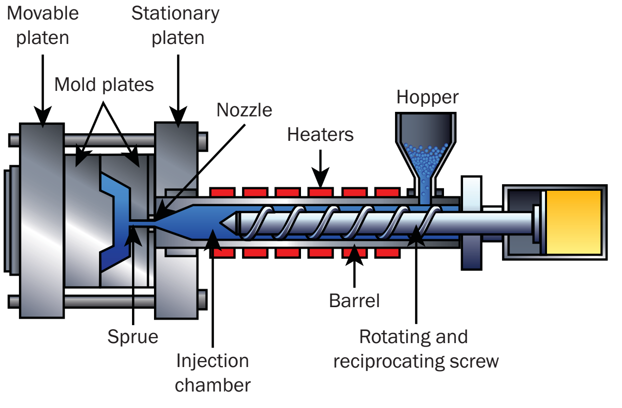


Figure A.2 A single screw injection-molding machine for thermoplastics, showing the plasticizing screw, a barrel, band heaters to heat the barrel, a stationary platen, and a movable platen

B.2.1.1 The Hopper

Thermoplastic material is supplied to molders in the form of small pellets. The hopper on the injection molding machine holds these pellets. The pellets are gravity-fed from the hopper through the hopper throat into the barrel and screw assembly.

B.2.1.2 The Barrel

As shown in Figure A.2, the barrel of the injection molding machine supports the reciprocating plasticizing screw. It is heated by the electric heater bands.

B.2.1.3 The Reciprocating Screw

The reciprocating screw is used to compress, melt, and convey the material. The reciprocating screw consists of three zones, illustrated in Figure A.3:

- Feeding zone
- Transition (or compressing) zone
- Metering zone

While the outside diameter of the screw remains constant, the depth of the flights on the reciprocating screw decreases from the feed zone to the beginning of the metering zone. These flights compress the material against the inside diameter of the barrel, which creates viscous (shear) heat. This shear heat is mainly responsible for melting the material. The heater bands outside the barrel help maintain the material in the molten state. Typically, a molding machine can have three or more heater bands or zones with different temperature settings.

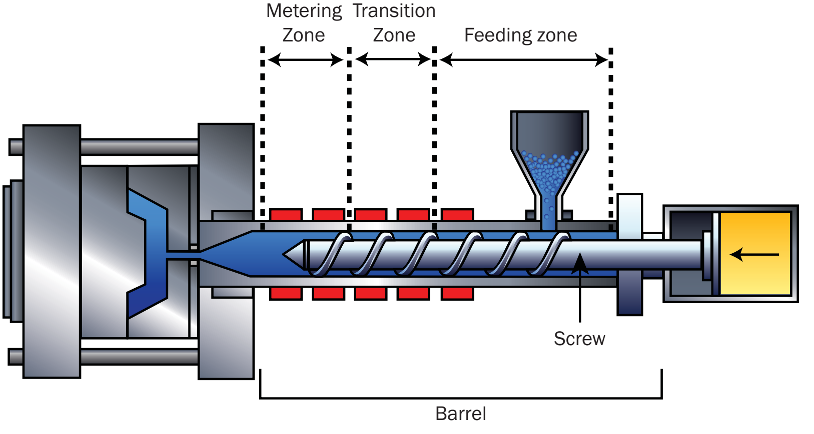


Figure A.3 A reciprocating screw, showing the feeding zone, transition (or compressing) zone, and metering zone

B.2.1.4 The Nozzle

The nozzle connects the barrel to the sprue bushing of the mold and forms a seal between the barrel and the mold. The temperature of the nozzle should be set to the material's melt temperature or just below it, depending on the recommendation of the material supplier. When the barrel is in its full forward processing position, the radius of the nozzle should nest and seal in the concave radius in the sprue bushing with a locating ring, as shown in Figure A.4 (a). During purging of the barrel, the barrel backs out from the sprue, as shown in Figure A.4 (b), so the purging compound can free fall from the nozzle.

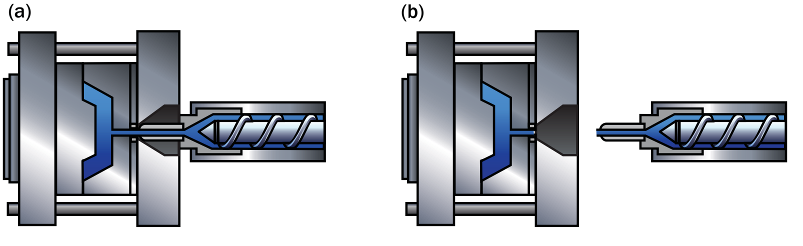


Figure A.4 (a) Nozzle with barrel in processing position (b) Nozzle with barrel backed out for purging

B.2.2 Mold System

The mold system consists of tie bars, stationary and moving platens, in addition to molding plates (bases), which house the cavity, sprue and runner systems, ejector pins, and cooling channels, as shown in Figure A.5. The mold is essentially a heat exchanger in which the molten thermoplastic solidifies to the desired shape and dimensional details defined by the cavity.

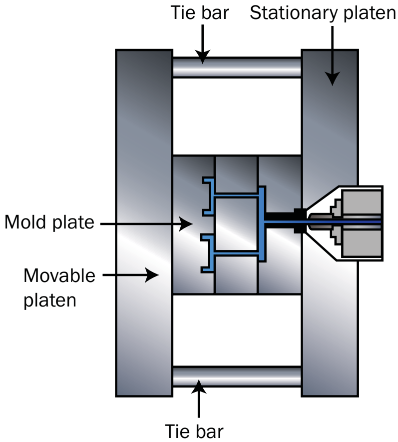


Figure A.5 A typical (three-plate) molding system

A mold system is an assembly of platens and molding plates typically made of tool steel. The mold system shapes the plastics inside the mold cavity (or matrix of cavities) and ejects the molded part(s). The stationary platen is attached to the barrel side of the machine and is connected to the moving platen by the tie bars. The cavity plate is generally mounted on the stationary platen and houses the injection nozzle. The core plate moves with the moving platen guided by the tie bars. Occasionally, the cavity plate is mounted to the moving platen and the core plate and a hydraulic knock-out (ejector) system is mounted to the stationary platen.

B.2.2.1 Two-plate Mold

The vast majority of molds consist essentially of two halves, as shown in Figure A.6 (a). This kind of mold is used for parts that are typically gated on or around their edge, with the runner in the same mold plate as the cavity.

B.2.2.2 Three-plate Mold

The three-plate mold is typically used for parts that are gated away from their edges. The runner is in between two plates, separate from the cavity and core, as shown in Figure A.6 (b).

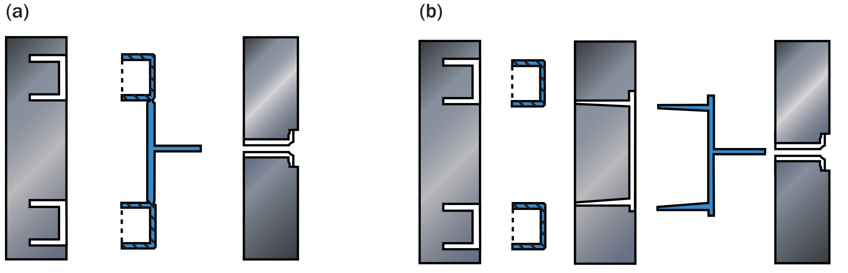


Figure A.6 (a) Two-plate mold (b) Three-plate mold

B.2.2.3 Cooling Channels (Circuits)

Cooling channels are passageways located within the body of a mold, through which a cooling medium (typically water, steam, or oil) circulates. Their function is the regulation of temperature on the mold surface. Cooling channels can also be combined with other temperature control devices, like bafflers, bubblers, and thermal pins or heat pipes.

B.2.3 Hydraulic System

The hydraulic system on the injection-molding machine provides the power to open and close the mold, build and hold the clamp tonnage, turn the reciprocating screw, drive the reciprocating screw, and energize ejector pins and moving mold cores. A number of hydraulic components are required to provide this power, including pumps, valves, hydraulic motors, hydraulic fittings, hydraulic tubing, and hydraulic reservoirs.

B.2.4 Control System

The control system provides consistency and repeatability in machine operation. It monitors and controls the processing parameters, including the temperature, pressure, injection speed, screw speed and position, and hydraulic position. The process control has a direct impact on the final part quality and the economics of the process. Process control systems can range from a simple relay on/off control to an extremely sophisticated microprocessor-based closed-loop control.

B.2.5 Clamping System

The clamping system opens and closes the mold, supports and carries the constituent parts of the mold, and generates sufficient force to prevent the mold from opening. Clamping force can be generated by a mechanical (toggle) lock, hydraulic lock, or a combination of the two basic types.

B.3 Molded System

A typical molded system consists of the delivery system and the molded part(s), as shown in Figure A.7.

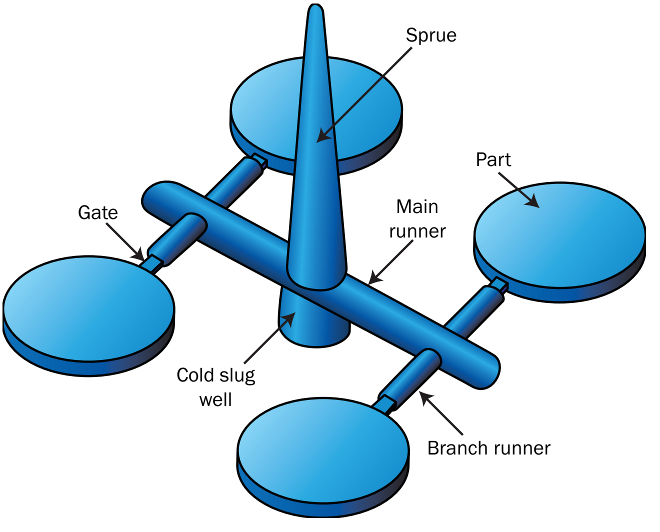


Figure A.7 The molded system includes a delivery system and molded parts.

B.3.1 The Delivery System

The delivery system, which provides passage for the molten plastic from the machine nozzle to the part cavity, generally includes:

- Sprue
- Cold slug wells
- Main runner
- Branch runners
- Gates

The delivery system design has a great influence on the filling pattern and thus the quality of the molded part.

B.3.1.1 Cold Runners

After molding the cold-runner delivery system is trimmed off and recycled. Therefore, the delivery system is normally designed to consume minimum material, while maintaining the function of delivering molten plastic to the cavity in a desirable pattern.

B.3.1.2 Hot Runners

The hot-runner (or runnerless) molding process keeps the runners hot to maintain the plastic in a molten state at all times. Since the hot-runner system is not removed from the mold with the molded part, it saves material and eliminates the secondary trimming process. The volume of the hot runner should be less than the shot size.

B.4 Machine Operating Sequence

Injection molding is a cyclic process. During the injection-molding process, the machine undertakes a sequence of operations in a cyclic fashion. A process cycle is one complete operation of an injection-molding machine.

B.4.1 Process Cycle

The basic injection-molding machine operations are shown in Figure A.8.

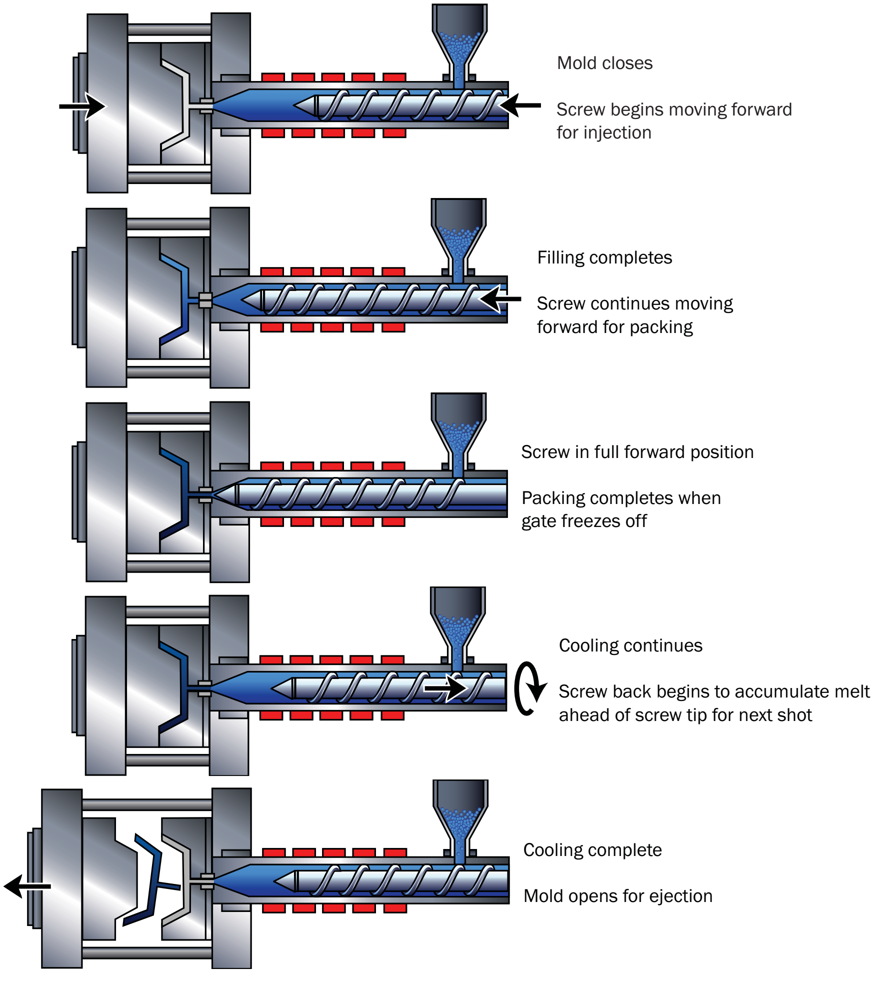


Figure A.8 The basic injection-molding machine operations

B.4.1.1 Cycle Time

Typical process cycle time varies from several seconds to tens of seconds, depending on the part weight, part thickness, material properties, and the machine settings specific to a given process.

B.5 Screw Operation

The reciprocating screw is used to plasticize the plastic pellets using various screw rotation speeds (revolutions per minute, or RPMs), inject the molten plastics as a plunger at various speeds and shot volumes, and control the pressure level in the molten plastic charge in front of the screw. Several of its operations are discussed here.

B.5.1 Back Pressure

Back pressure is the amount of pressure exerted on the material volume ahead of the screw as the screw is pushed back in preparation for the next shot.

B.5.1.1 Setting the Maximum Back Pressure

Typically, all machines have an adjustment for the maximum back pressure. This *screw-back* stage stops when the screw reaches a preset position. The stop position is manually set, based on the amount of material required to fill the mold's cavity and runner system. When the machine is ready to inject the shot, the screw then plunges the material ahead of the screw forward, injecting it into the mold. While the injected material is cooled in the mold after the injection, the screw-back stage is re-initiated and the molding cycle repeated.

B.5.2 Injection Speed

The injection (or ram) speed is the forward speed of the screw during its injection operation.

B.5.2.1 Setting the Injection Speed

For most engineering resins, the ram speed should be set to the fastest setting that the part design and process will allow for technical and economic reasons. However, slower injection speed at the beginning of injection may be necessary to avoid turbulent flow and jetting, as material passes through the restrictive areas (e.g. gates). The injection speed should be reduced again toward the end of injection to avoid flashing at the end of stroke, and to enhance the formation of homogenous weld lines after a divided flow.

B.5.3 Screw Rotation Speed

The screw rotation speed (RPM) is the rate at which the plasticizing screw rotates. The faster the screw rotates, the faster the material is compressed by the screw flights, increasing the amount of shear heating.

B.5.4 Cushion

The cushion is the difference in the final forward position of the screw and its maximum allowable forward position. If the screw were allowed to travel its full stroke and stop mechanically against the nozzle, the cushion would be zero. Typically a cushion of 3 to 10 mm (1/8 to 3/8 in) is used.

B.6 Secondary Operations

After a part is ejected, the delivery system (sprue, runners, and gates) is trimmed off as a secondary operation. For some applications, additional secondary operations are needed for assembly or decoration. Detailed descriptions of these secondary operation procedures can be found in design handbooks from material suppliers.

B.6.1 Assembly

Secondary operations for assembling parts include:

- Bonding
- Welding
- Inserting
- Staking
- Swaging
- Assembling with fasteners

B.6.2 Decoration

Secondary operations for decorating the plastic parts include:

- Applique: a surface covering applied by heat and pressure
- Printing: a process of making a mark or impression onto a substrate for decorative or informational purposes.

B.6.3 Other Secondary Operations

Other secondary operations include:

- Painting
- Hard coating
- Metallizing/shielding
- Surface treatment
- Annealing
- Machining

Appendix C: Injection-molding Process Control

- Importance of process conditions
- Setting machine process conditions

C.1 Importance of Process Conditions

The quality of the molded part is greatly influenced by the conditions under which it is processed. See, for example, the process window shown in Figure A.1. As you lower the temperature, higher pressure is needed to deliver the polymer melt into the cavity. If the temperature is too high, you risk causing material degradation. If the injection pressure is too low, a short shot could result. If the pressure is too high, you will flash the mold.

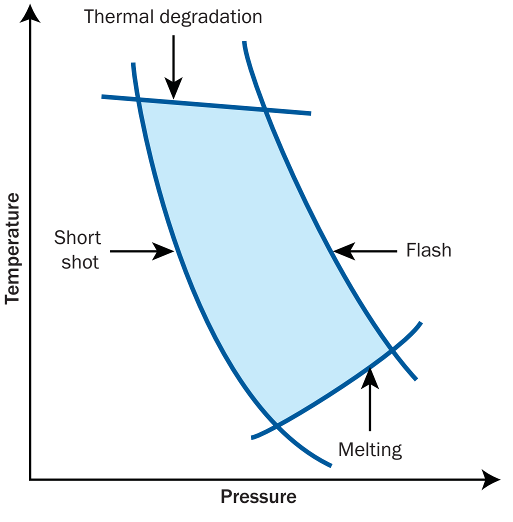


Figure A.1 Process window showing the influence of pressure versus temperature

C.1.1 Setting Machine Process Conditions

Before setting process conditions, you should make certain the molding machine is in proper working order, and that the mold you plan to use was designed for the particular machine you plan to use. Follow the step-by-step procedure provided below to control the settings on your machine.

C.1.1.1 Step 1— Set the Melt Temperature

Melt temperature is one of the most important factors in molding plastic parts. If it is too low, the resin might not be completely melted or it might be too sticky to flow. If the melt temperature is too high, the resin could degrade, especially if the resin is POM or PVC. Suggested melt and mold temperatures for specific materials are available from the resin supplier. Appendix D contains a reference list of resins, their general properties, and typical applications. Appropriate melt and mold temperatures for several materials are listed in Table 2.

Setting Heater Band Temperatures: Most melting of the resin occurs because of the frictional heating from the screw rotation inside the barrel. The barrel heater bands serve mainly to keep the resin at the appropriate temperature. Typically there are three to five temperature zones or heater bands on the cylinder. The rules for setting the heater band temperatures are as follows:

- The last temperature zone, nearest the hopper, should be about 40 to 50ºC (72 to 80ºF) lower than the calculated melt temperature to give better transport of plastic pellets during plasticization.
- The heater band at the nozzle zone should be set to the calculated melt temperature, and should keep the temperature uniform. Improper heater band temperature settings may cause drooling at the nozzle, and degradation or color change, especially for PA materials.

Air-shot Temperature: The actual melt temperature, or air-shot temperature, is usually higher than the heater band controller setting. This difference is because of the influence of back pressure and screw rotation on frictional heating and the melt temperature, as mentioned above. (You can measure the actual melt temperature by quickly sticking a probe thermometer into an air shot with the nozzle backed away from the mold.)

C.1.1.2 Step 2—Set the Mold Temperature

Suggested melt and mold temperatures for specific materials are available from the resin supplier. Appropriate melt and mold temperatures for many generic, base resins are listed in Table 2. The mold temperature can be measured by using a thermometer. As illustrated in Figure A.2, the average cavity surface temperature will be higher than the temperature of the coolant during production. Thus, you should set the coolant temperature to be 10 to 20ºC (18 to 36ºF) lower than the required mold temperature. If the mold temperature is 40 to 50ºC (72 to 80ºF) or more, consider insulation plates between the mold and the clamping plates for energy savings and process stabilization.

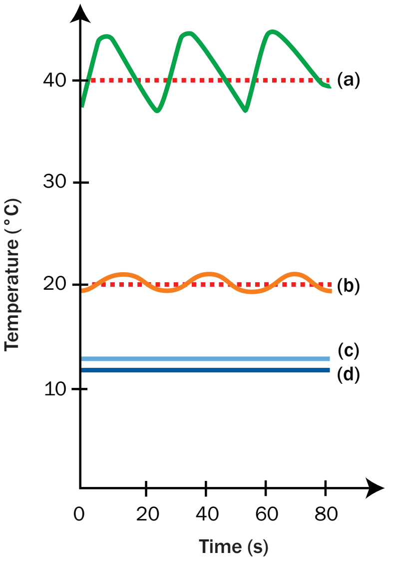


Figure A.2 Temperature-time curve at various locations in the mold (a) Mold cavity surface (b) Cooling channel wall (c) Cooling channel outlet (d) Cooling channel inlet

Use the lowest temperature setting to achieve the shortest cycle time. However, you might try using higher temperatures to improve the appearance of the part. A higher mold temperature produces a higher gloss and more crystallization.

Considering Temperature Difference: For parts with a deep core, a lower coolant temperature is needed for the core (moving plate) in order to minimize the temperature difference between the mold surfaces on the core and cavity. A lower surface temperature difference will produce parts with higher quality, at a lower cost. By a rule of thumb, the coolant temperature for fixed and moving plates should not differ by more than 20°C (36°F). This is related to thermal expansion, which can be determined only by the user. A large temperature difference results in differential mold plate thermal expansion, which may cause alignment problems in guide pins, especially in large molds. The mold will sometimes lock up for this reason. The cycle time can be increased to reduce the required coolant temperature difference.

C.1.1.3 Step 3—Set the Switch-over Position

The switch-over position is the ram position where the velocity controlled filling (injection) stage switches to the pressure controlled packing phase. Once the ram is under pressure control, the ram continues to move forward to pressurize the cavity and to compensate for shrinkage. The cushion is the distance from the ram position at the end of the packing phase to the farthest position that the end of the screw can reach, as shown in Figure A.3. The typical cushion distance is about 3 to 10 mm (1/8 to 3/8 in).

At this step, set the switch-over position to fill about two-thirds of the mold. This prevents damage to the press or the mold. In Step 14, the injection volume will be increased to fill 95% of the cavity.

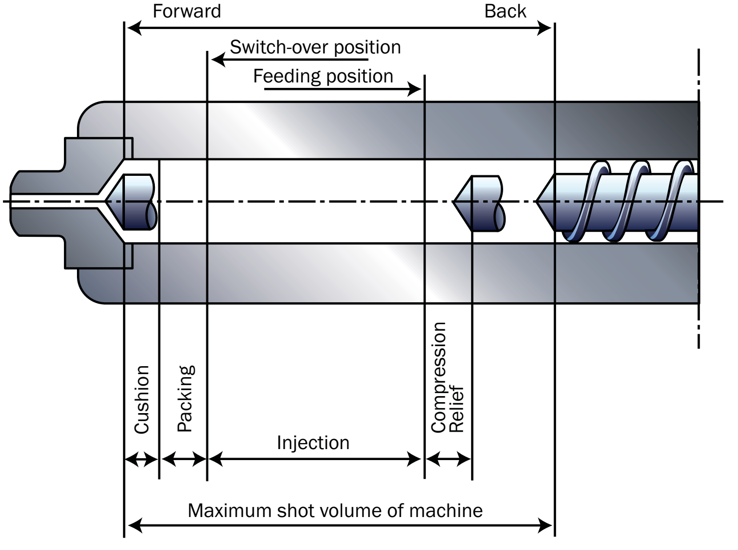


Figure A.3 Screw positions at various stages

C.1.1.4 Step 4—Set the Screw Rotation Speed

Set the screw rotation speed to the level required to plasticize the resin. Plasticizing should not prolong the cycle time. If it does, increase the speed. The ideal speed causes plasticizing to complete at the latest possible point in the cycle without exceeding cooling time and prolonging the cycle time. Resin vendors supply the suggested screw rotation speed for specific resins.

C.1.1.5 Step 5—Set the Back Pressure

The recommended back pressure is about 5 to 10 MPa. Back pressure that is too low can result in inconsistent parts. Increasing the back pressure will increase the frictional contribution to the melt temperature and decrease the plasticization time. To speed up plasticization, use a higher back pressure to achieve a shot volume that is a larger percentage of the injection machine's capacity. Use a lower back pressure for a smaller percentage shot volume because the material will remain in the barrel longer (for many cycles) before it reaches the screw head.

C.1.1.6 Step 6—Set the Injection Pressure to the Machine Maximum

The injection pressure is the pressure of the melt in front of the screw. The injection pressure should be as low as possible to reduce part internal stress. On the machine, set the injection pressure to nearly the machine maximum. The purpose is to completely exploit the injection velocity of the machine, so that the pressure setting valve does not limit the velocity. Because the switch-over to packing pressure occurs before the mold is completely filled, no damage will be done to the mold.

C.1.1.7 Step 7—Set the Packing Pressure at 0 Mpa

For now, set the packing pressure at 0 MPa, so the screw will stop when it reaches the switch-over position. This will prevent mold or press damage. In Step 17, the packing pressure is increased to its final setting.

C.1.1.8 Step 8—Set the Injection Velocity to the Machine Maximum

With the highest possible injection velocity within shear rate limits, you can expect less flow resistance, longer flow length, and improved strength in weld lines. However, you may need to create additional vents once you do this.

Proper Venting Minimizes Defects: Insufficient venting causes compression of air trapped in the cavity. This results in very high temperatures and pressures in the cavity, causing burn marks, material degradation, and short shots. You should design a venting system to avoid or minimize the defects caused by trapped air in the mold. Moldflow shows you where weld lines, meld lines, and air trap locations will occur: use these predictions to improve your design. Remember that it is necessary to clean the mold surface and venting system regularly, especially for PVC or ABS/PVC materials.

C.1.1.9 Step 9—Set the Packing Time

The ideal packing time setting is the gate freezing (sealing) time or the part freezing time, whichever is shorter. The gate and part freezing times can be calculated or estimated. The calculated values for the packing time are based on packing analysis results when the frozen layer fraction is 1.0 for the gate. Without packing analysis results, the packing time is estimated to be 10 times the filling time.

C.1.1.10 Step 10—Set Ample Cooling Time

Cooling time can be calculated or estimated. The cooling time is after packing time, as shown in Figure A.4. During cooling the part continues to solidify so it can be ejected, and material for the next shot is prepared. The calculated value of cooling time is from a cooling or packing analysis. Without Moldflow results, the cooling time can be 10 times the filling time. For example, if the predicted filling time is 0.85 seconds, the initial cooling time would be 8.5 seconds. The combination of packing time (if estimated would be 8.5 seconds) and the cooling time should be a high estimate to ensure the part and runner system will be sufficiently solid for ejection.

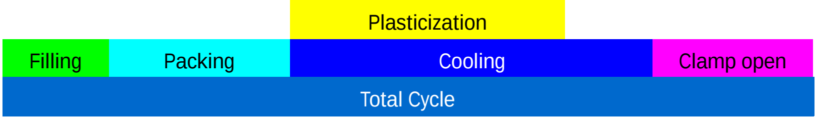


Figure A.4 Cycle time and its components

C.1.1.11 Step 11—Set the Mold Opening Stroke

The mold opening stroke is comprised of the core height, the part height, and the capsize space, as shown in Figure A.5. You should minimize the mold opening stroke. The mold opening speed should be slow at the very beginning, then accelerate, then slow down again at the end of the stroke. The sequence of the mold closing speed is similar to the mold opening speed: slow, fast, slow.

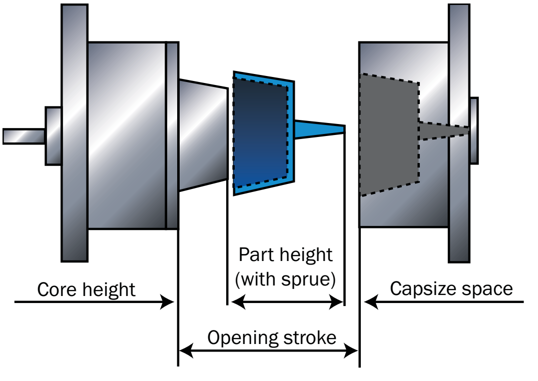


Figure A.5 Required mold opening

C.1.1.12 Step 12—Set the Ejector Stroke, Start Position, and Velocity

Relieve any slides first. The ejector travel should be, at a maximum, the core height. If the machine is equipped with a hydraulic ejector, set the start position at the point where the part is clear of stationary mold parts. (When the ejector velocity is equal to the opening speed, the part remains where it was in relation to the stationary mold part.)

C.1.1.13 Step 13—Set the Mold Open Time

The mold open time is usually set at 2 to 5 seconds. This includes mold opening, ejection of parts from the mold, then mold closing, as shown in Figure A.4. The cycle time is the sum of the filling time, cooling time, and mold open time.

C.1.1.14 Step 14—Mold a Short-shot Series by Increasing Injection Volume

Moldflow provides the part weight and sprue/runner/gate weight. From this information, along with the screw diameter or barrel inner diameter, the total injection volume and the feeding position (see Figure A.3) can be estimated for each shot.

For now, fill only two-thirds of the mold. The packing pressure should already be set at 0 MPa, so that mold filling stops when the screw reaches the switch-over position, thus protecting the mold structure and the press. Next, increase the volume in increments of 5 to 10%, up to 95% of mold filling.

In order to prevent material from escaping from the open nozzle, relieve the back pressure created during plasticizing by drawing back the screw a few millimeters, immediately after the rotation has stopped.

C.1.1.15 Step 15—Switch to Automatic Operation

The purpose of an automatic operation is to obtain stability in the process.

C.1.1.16 Step 16—Set the Injection Volume to 99% Mold Filled

When the process has stabilized (when the same parts are produced each time), adjust the switch-over position to 99% of filling. This will exploit the maximum injection speed in as large a part of the injection as possible.

C.1.1.17 Step 17—Increase the Packing Pressure in Steps

Increase the packing pressure in steps of approximately 10 MPa in the melt. If the first step does not fill the mold completely, increase the injection volume.

De-mold and remove the part. Write the packing pressure on it. This packing pressure series forms a good basis for a more thorough examination. You can then discuss the possibilities and limitations with the customer.

Choose the lowest acceptable packing pressure, as this minimizes the internal stresses in the part and saves material, as well as operating costs. A high packing pressure can cause excessive residual stresses that could warp the part. Molded-in residual stresses can be released somewhat by annealing at around 10°C (18°F) below the heat deflection temperature.

If the material cushion is completely used (see Figure A.3), the last part of the packing pressure time will not be effective. This calls for a change in the injection stroke position, in order to increase the injection volume.

Calculating Injection Pressure: The hydraulic pressure in the injection cylinder can be read on the machine manometer. However, the injection pressure in front of the screw is more important. To calculate the injection pressure you will need to multiply the Intensification ratio (hydraulic pressure by the resin/hydraulic pressure ratio). This ratio is usually found on the molding machine near the injection unit or in the instruction manual for the machine. The ratio is usually in the range of 7 to 15, as shown in Figure A.6 below.

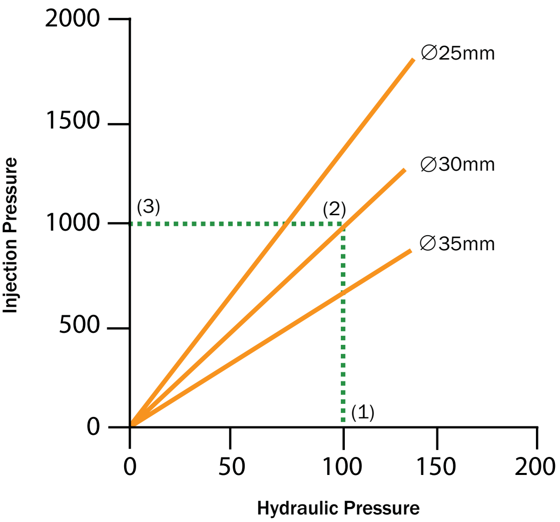


Figure A.6 Intensification ratio for a ∅ 30 mm screw is 11.1

C.1.1.18 Step 18—Minimize the Packing Time

If consistent part dimensions are essential, the following method of packing time determiniatin is more accurate. The gate seal time must be determined. The gate seal time can be determined experimentally on the molding machine with a gate seal study. This involves initially starting with a long packing time and reducing the pack time until the part weights begin to change. This is an indication that the gate is open. When the packing time is decreased, the cooling time should increase the same amount to maintain the same cycle time. For example, Figure A.7 shows that the packing pressure does not influence the part weight after 9 seconds. This is your minimum packing time.

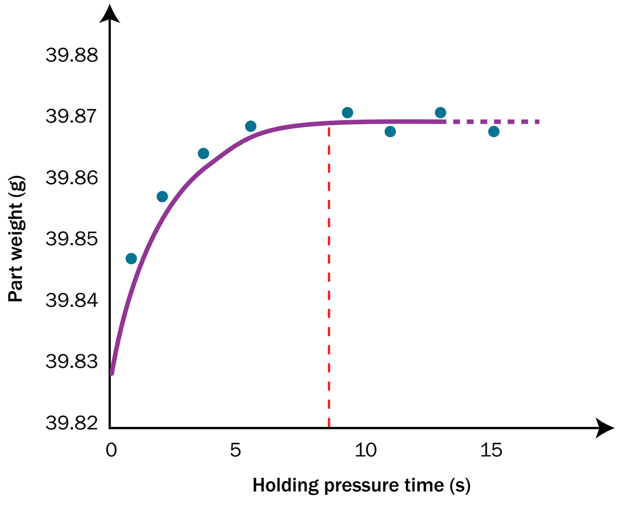


Figure A.7 Determination of the gate/part freezing time by weighing parts manufactured at various packing times

C.1.1.19 Step 19—Minimize the Remaining Cooling Time

Reduce the remaining cooling time until the maximum surface temperature of the part reaches the heat deflection temperature of the material. The heat deflection temperature can be provided by the resin supplier.

Appendix D: Plastic Materials

- What are plastics?
- Classification of plastics
- Thermoplastics
- Thermosets
- Properties and applications of thermoplastics
- Additives, fillers, and reinforcements

D.1 What Are Plastics?

D.1.1 Polymerization Process

Plastics are one group of polymers built from relatively simple units called monomers (or mers) through a chemical polymerization process. This process is illustrated in Figure A.8. Processing polymers into end products mainly involves physical phase change, such as melting and solidification (for thermoplastics) or a chemical reaction (for thermosets).

D.1.2 Structure of Polymers

The basic structure of a polymer molecule can be visualized as a long chain of repeating units, with additional chemical groups forming pendant branches along the primary "backbone" of the molecule, as shown in Figure A.8. Although the term *plastics* has been loosely used as a synonym for polymer and resin, plastics generally represent polymeric compounds that are formulated with plasticizers, stabilizers, fillers, and other additives for purposes of processibility and performance. Other polymeric systems include rubbers, fibers, adhesives, and surface coatings. A variety of processes have been employed to produce the final plastic parts, as illustrated in Figure A.9.

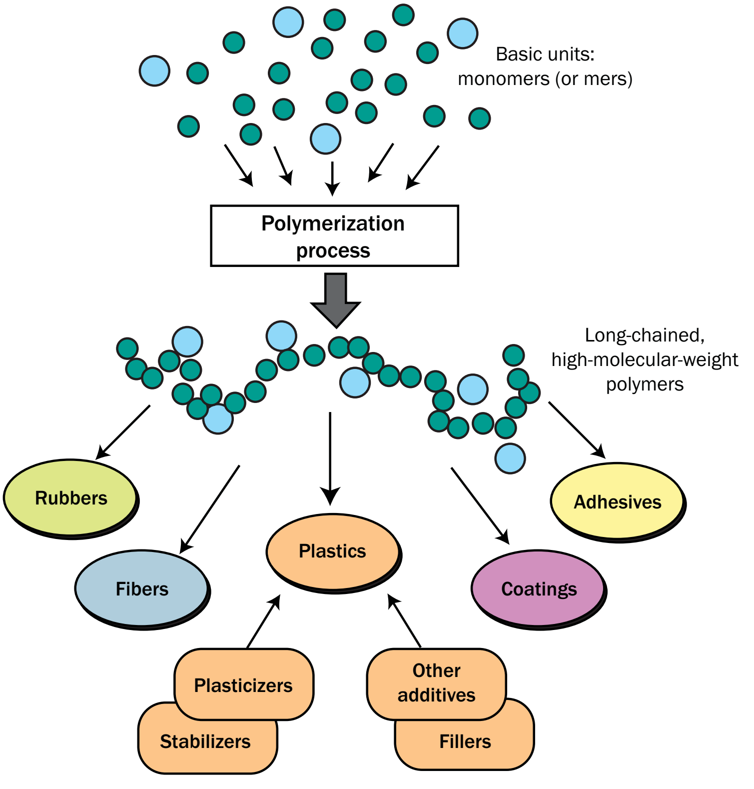


Figure A.8 Polymer family: the formation of plastics and the polymerization process

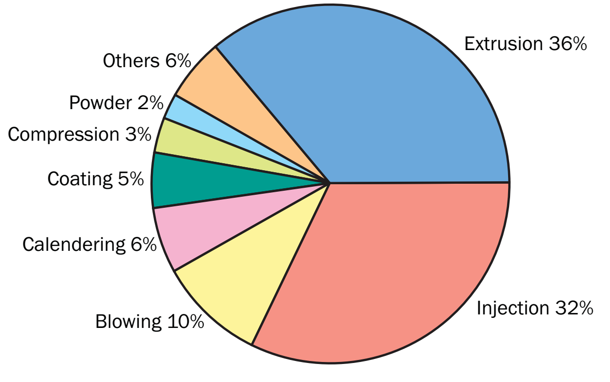


Figure A.9 Plastics consumption by process

D.1.2.1 Structure-dependent Properties

The structural arrangement, size, and chemical constitution of the polymer molecule have a direct influence on its physical and chemical properties. In addition, the macromolecular nature of plastics implies that their material properties may also be dependent on the mechanical and thermal history that the materials experience during processing. For example, the viscosity (which indicates the material's resistance to flow) of a polymer melt increases with increasing molecular weight, but decreases as temperature increases. Further, the aligned molecular orientation that results from strong shear exerted on the material also reduces the viscosity of the polymer melt.

The physical and mechanical properties, as well as the cost of polymers, can be modified by blending a number of polymers or by compounding them with other materials or reinforcing agents. These processes have resulted in the following polymeric systems.

D.1.3 Polymer Alloys and Blends

Polymer alloys and blends are mixed systems of two or more finished polymers. When the combination of polymers has a single glass transition temperature and yields a synergistic effect (i.e., the properties of the mix are better than either of the individual components), the resulting system is termed a polymer alloy. When the resulting product has multiple glass transition temperatures and its properties are the average of the individual components, the material system is referred to as a polymer blend. One of the earliest commercially successful blends was ABS, which combines the chemical resistance, toughness, and rigidity of its components.

D.1.4 Polymer Composites

Polymer composites are materials that incorporate certain reinforcing agents into a polymer matrix to add desirable properties. Low aspect ratio materials—such as single crystal/whisker, and flake-type fillers of clay, talc, and mica—impart increased stiffness. On the other hand, larger aspect ratio reinforcements—such as fibers or filaments of glass, carbon-graphite, aramid/organic, and boron—substantially raise both the tensile strength and the stiffness.

D.2 Classification of Plastics

Based on the type of chemical reaction (polymerization) that links the molecules together, plastics are classified as either thermoplastics or thermosets.

D.2.1 Classes of Plastics

In addition to the broad categories of thermoplastics and thermosets, thermoplastics can be further categorized into amorphous, (semi-)crystalline, or liquid crystal polymers (LCPs), depending on the polymer chain conformation or morphology. The microstructures of these plastics and the effects of heating and cooling on the microstructures are shown in Figure A.10. Other classes include elastomers, copolymers, compounds, commodity resins, and engineering resins. Additives, fillers, and reinforcements are other classifications that relate directly to plastics properties and performance.

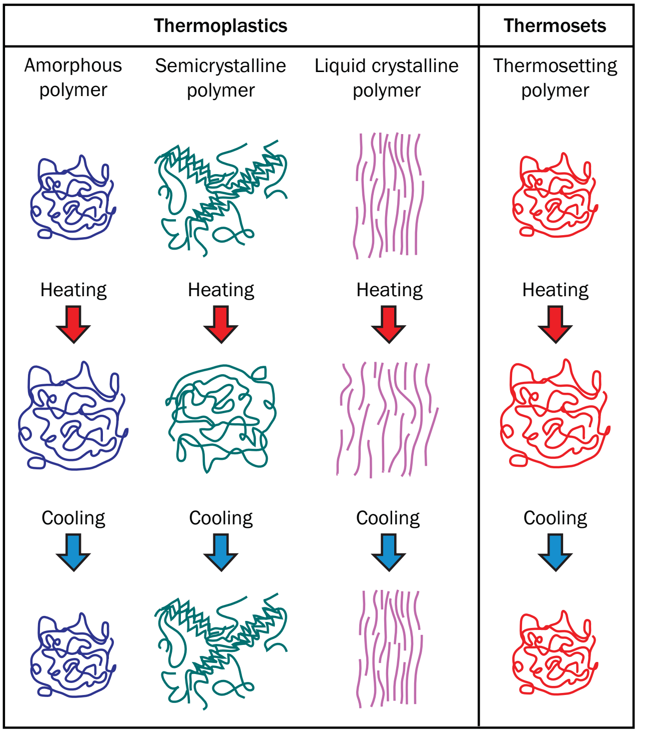


Figure A.10 Microstructure of various plastics and effect of heating and cooling during processing

D.2.2 Structures and Properties of Plastics

Table A.1 lists a summary of the relevant structures and properties of thermoplastics and thermosets.

Table A.1: Structures and properties of thermoplastics and thermosets

	Thermoplastics	Thermosets
Microstructure	Linear or branch molecules No chemical bonds among the molecules	Cross-linking network with chemical bonds among molecules after the chemical reaction
Reaction to heat	Can be re-softened (physical phase change)	Cannot be re-softened after cross-linking without degradation
General properties	Higher impact strength Easier processing Better adaptability to complex shapes	Greater mechanical strength Greater dimensional stability Better heat and moisture resistance.

D.3 Thermoplastics

Thermoplastics typically have high molecular weights resulting from a high degree of polymerization. The long molecular chain, either linear or branched, has side chains or groups that are not attached to other polymer molecules. As a result, thermoplastics can be repeatedly softened (or hardened) by an increase (or decrease) in temperature. This type of phase change without a chemical reaction permits the recycling of thermoplastic scraps, such as the trimmed-off runners and sprues from injection molding. An analogy is the phase change of ice turning into water under heat, and then becoming a solid again when cooled. Although thermoplastics are recyclable, it is very likely that a small degree of chemical change (e.g., oxidation or thermal degradation) takes place during processing, and therefore the properties of recycled polymers may not be equivalent to those of the virgin polymer.

D.3.1 Market Share Distribution of Thermoplastics

Thermoplastics account for more than 70% of all polymers produced. Thermoplastic materials are purchased as pellets or granules. They are melted by heat under pressure into a relatively viscous fluid and shaped into a desirable product or form by cooling. Thermoplastics generally offer higher impact strength, easier processing, and better adaptability to complex designs than do thermosets.

D.3.1.1 Commodity Resins

Among thermoplastics, the commodity resins—for example, high-density polyethylene (HDPE), low-density polyethylene (LDPE), polypropylene (PP), polystyrene (PS), and polyvinyl chloride (PVC)—account for more than 90% of all thermoplastics.

D.3.1.2 Engineering Resins

On the other hand, the engineering resins—such as acetal, acrylonitrile butadiene styrene (ABS), nylon, and polycarbonate (PC)—offer improved performance, including higher mechanical properties, better heat resistance, and higher impact strength. Thus, they demand a higher price.

D.3.2 Structures and Properties of Thermoplastics

Table 1 lists a summary of the relevant structures and properties of amorphous polymers and crystalline polymers.

Table 1: Structures and properties of amorphous and crystalline polymers

	Amorphous Polymers	Crystalline Polymers
Common materials	Acrylonitrile butadiene styrene (ABS) Acrylics (e.g., PAN, PMMA) Polycarbonate (PC) Polystyrene (PS) Polyvinyl chloride (PVC) Styrene acrylonitrile (SAN)	Acetals Nylon Polyethylene (PE) Polypropylene (PP) Thermoplastic Polyesters (e.g., PBT, PET)
Microstructure	Random molecular orientation in both molten and solid phases.	Random molecular orientation in molten phase, but densely packed crystallites occurs in solid phase.
Reaction to heat	Softens over a range of temperatures (no apparent melting temperature)	Fairly distinct melting temperature.
General properties	Transparent Poor chemical resistance Low volumetric shrinkage in molding Generally low strength Generally high melt viscosity Lower heat content	Translucent or opaque Excellent chemical resistance High volumetric shrinkage in molding Generally high strength Generally low melt viscosity Higher heat content (with heat of crystallization)

D.3.3 Amorphous Polymers

Molten polymer molecules in an unstressed state are randomly oriented and entangled with other molecules. Amorphous materials retain this type of entangled and disordered molecular configuration regardless of their states, as shown in Figure A.10.

D.3.3.1 Response to Temperature

When the temperature of melt decreases, amorphous polymers start becoming rubbery. When the temperature is further reduced to below the glass transition temperature, the amorphous polymers turn into glassy materials. Amorphous polymers possess a wide softening range (with no distinct melting temperature), moderate heat resistance, good impact resistance, and low shrinkage.

D.3.3.2 Differential Shrinkage in Thickness Direction

The molecules tend to be uncoiled and stretched in the flow direction as the cavity is filled. Those molecules that are quenched by contact with the cold mold wall will be frozen, stretched out in the flow direction. Molecules toward the interior of the part are insulated from the mold wall by the frozen layer. These will have time enough to recoil as they cool more slowly. That is, the molecules on the surface will be oriented and will shrink less; molecules in the interior will be less oriented and will shrink more. The differential shrinkage in the thickness direction results in flow-induced residual stresses in molded plastics.

D.3.3.3 Similar Linear Shrinkages

Families of amorphous plastics can often be substituted one for another, in the same injection cavities, since their linear shrinkages are in the same range. Therefore, styrene can be substituted for ABS; acrylics can be molded in the same cavities as polycarbonates. The properties will be different for the substitution, but the dimensions will usually be close enough to be within specified tolerances.

D.3.4 Semicrystalline Polymers

Crystalline materials are polymer chains that do not have bulky pendant groups, chain branches, or cross-links. They may accommodate themselves in a well-ordered regular lattice (polymer crystallite) when the molten polymers are cooled below the melting temperature, as shown in Figure A.10.

D.3.4.1 Controlling the Degree of Crystallinity

The crystallization process stops when the materials are cooled below the glass transition temperature. Since it is difficult to achieve 100% crystallization under normal processing conditions, any crystallizable polymers are typically semicrystalline, possessing both amorphous and crystalline phases. The degree of crystallinity depends on both the chemical structure of the polymer and the processing conditions. Crystalline and semicrystalline polymers have a distinct melting point, good chemical and heat resistance, good lubricity, low moisture absorption, and high shrinkage.

D.3.4.2 High Linear Shrinkage

The significantly higher linear shrinkage of the semicrystalline polymers precludes them being molded in the same cavities that are used for amorphous plastics: most dimensions will be significantly different and will most likely miss tolerances enough not to be functional in the same application.

D.3.4.3 Liquid Crystal Polymers

Liquid crystal polymers (LCPs) exhibit ordered molecular arrangements in both the melt and solid states, as shown in Figure A.10. These materials are characterized by their stiff, rodlike molecules that form the parallel arrays or domains. LCPs offer a number of processing and performance advantages including low melt viscosity, low mold shrinkage, chemical resistance, stiffness, creep resistance, and overall dimensional stability.

D.4 Thermosets

Cross-linking is a chemical process in which chemical bonds form among molecules of thermosetting materials, resulting in an interconnected network, as shown in Figure A.10. This cross-linking process is the principal difference between thermoplastics and thermosets. Thermosets inherently possess greater mechanical strength, higher service temperature limits, and greater dimensional stability than thermoplastics. Many thermosets are engineering resins and, because of the cross-linking, thermosets possess an amorphous structure.

D.4.0.1 Cross-linking (Reaction)

Prior to molding, the chainlike structure of thermosets is similar to thermoplastics. During processing, thermosets polymerize (react or cure) with the activation of heat and/or a chemical means into a cross-linked microstructure. Once the reaction is completed, the polymer chains are bonded (cross-linked) together to form a three-dimensional network. These cross bonds among molecules prohibit the slippage of individual molecular chains. Consequently, a thermoset becomes an infusible and insoluble solid and cannot be re-softened and reprocessed through the application of heat, without degrading some linkages. The best analogy to thermosets is that of a hard-boiled egg; the yolk has turned from a liquid to a solid and cannot be converted back to a liquid.

D.4.0.2 Processing Thermosets

Thermosets are usually purchased as liquid monomer-polymer mixtures or as a partially polymerized molding compound. Starting from this uncured condition, they can be formed to the final shape in the cavity by polymerization (activated either by heat or by chemical mixing) with or without pressure. Thermosets are generally filled or reinforced with materials (such as minerals, talc, or glass fibers) to impart specific properties (such as shrinkage control, chemical and shock resistance, electrical and thermal insulation) and/or to reduce cost.

D.5 Properties and Applications of Thermoplastics

This section reviews the general properties and typical applications of common thermoplastic resins used for injection molding. Generally, only injection-molded engineering applications are listed; packaging and textiles, for example, are not included.

Table 2: Typical melt and mold temperatures for various generic classes of resins

Generic Name	Melt Temperature (C/F)			Mold Temperature (C/F)			Ejection Temp (C/F)
	Min.	Rec.	Max.	Min.	Rec.	Max.	Rec.
ABS	200/392	230/446	280/536	25/77	50/122	80/176	88/190
PA 12	230/446	255/491	300/572	30/86	80/176	110/230	135/275
PA 6	230/446	255/491	300/572	70/158	85/185	110/230	133/271
PA 66	260/500	280/536	320/608	70/158	80/176	110/230	158/316
PBT	220/428	250/482	280/536	15/60	60/140	80/176	125/257
PC	260/500	305/581	340/644	70/158	95/203	120/248	127/261
PC/ABS	230/446	265/509	300/572	50/122	75/167	100/212	117/243
PC/PBT	250/482	265/509	280/536	40/104	60/140	85/185	125/257
HDPE	180/356	220/428	280/536	20/68	40/104	95/203	100/212
LDPE	180/356	220/428	280/536	20/68	40/104	70/158	80/176
PEI	340/644	400/752	440/824	70/158	140/284	175/347	191/376
PET	265/509	270/518	290/554	80/176	100/212	120/248	150/302
PETG	220/428	255/491	290/554	10/50	15/60	30/86	59/137
PMMA	240/464	250/482	280/536	35/90	60/140	80/176	85/185
POM	180/356	210/410	235/455	50/122	70/158	105/221	118/244
PP	200/392	230/446	280/536	20/68	50/122	80/176	93/199
PPE/PPO	240/464	280/536	320/608	60/140	80/176	110/230	128/262
PS	180/356	230/446	280/536	20/68	50/122	70/158	80/176
PVC	160/320	190/374	220/428	20/68	40/104	70/158	75/167
SAN	200/392	230/446	270/518	40/104	60/140	80/176	5/185

D.5.1 ABS

D.5.1.1 Generic Class

Acrylonitrile-Butadiene-Styrene

D.5.1.2 Typical Applications

- Automotive (instrument and interior trim panels, glove compartment doors, wheel covers, mirror housings, etc.)
- Refrigerators, small appliance housings and power tools applications (hair dryers, blenders, food processors, lawn mowers, etc.)
- Telephone housings, typewriter housings, typewriter keys
- Recreational vehicles (golf carts, jet skis, etc.)

D.5.1.3 Injection-molding Processing Conditions

Drying: ABS resins are hygroscopic and drying is required prior to processing. Suggested drying conditions are 80–90°C (176–195°F) for a minimum of 2 hours. Resin moisture content should be less than 0.1%.

Melt Temperature: 200–280°C (392–536°F); Aim: 230°C (446°F).

Mold Temperature: 25–80°C (77–176°F). (Mold temperatures control the gloss properties; lower mold temperatures produce lower gloss levels).

Injection Pressure: 50–100 MPa (7,250–14,500 psi).

Injection Speed: Moderate to high.

D.5.1.4 Chemical and Physical Properties

ABS is produced by a combination of three monomers: acrylonitrile, butadiene, and styrene. Each of the monomers impart different properties: hardness, chemical and heat resistance from acrylonitrile; processability, gloss, and strength from styrene; and toughness and impact resistance from butadiene. Morphologically, ABS is an amorphous resin.

The polymerization of the three monomers produces a terpolymer that has two phases: a continuous phase of styrene-acrylonitrile (SAN) and a dispersed phase of polybutadiene rubber. The properties of ABS are affected by the ratios of the monomers and molecular structure of the two phases. This allows a good deal of flexibility in product design and, consequently, there are hundreds of grades available in the market. Commercially available grades offer different characteristics, such as medium to high impact, low to high surface gloss, and high heat distortion.

ABS offers superior processability, appearance, low creep and excellent dimensional stability, and high impact strength.

D.5.1.5 Major Manufacturers

- BASF (Terluran)
- LANXESS (Lustran)
- Cheil Synthesis (Starex)
- Chi Mei (Polylac)
- Dow Chemical (Magnum)
- GE Plastics (Cycolac)
- LG Chemical (Lupos)

D.5.2 PA 12

D.5.2.1 Generic Class

Polyamide 12 or Nylon 12.

D.5.2.2 Typical Applications

- Gear wheels for water meters and business machines
- Cable ties
- Cams
- Slides
- Bearings

D.5.2.3 Injection-molding Processing Conditions

Drying: The moisture content must be below 0.1% prior to processing.

If the material is exposed to air, drying in a hot air oven at 85°C (185°F) for 4 to 5 hours is recommended (3 to 4 hours in a desiccant dryer). If the container is unopened, it may be used directly for molding after 3 hours of equilibration to shop floor temperature.

Melt Temperature: 230–300°C (446–580°F); not to exceed 310°C (590°F) for standard grades and 270°C (518°F) for flame retardant grades.

Mold Temperature: 30–40°C (86–104°F) for unreinforced grades; for thin walled or large surface area components, 80–90°C (176–194°F) may be used; 90–100°C (194–212°F) for reinforced grades. Increasing the mold temperature increases the crystallinity level. It is very important to precisely control the mold temperature.

Injection Pressure: Up to 100 MPa (14, 500 psi). Low hold pressures and high melt temperatures are recommended.

Injection Speed: High (high speeds give a better finish on glass-filled grades).

D.5.2.4 Runners and Gates

Runner diameters for unfilled grades may be as small as 3 to 5 mm because of the material's low viscosity. Reinforced grades require larger diameters (5 to 8 mm). The runner shape should be the fully round type. Sprues should be as short as possible.

A variety of gates may be used. Small gates for large parts should not be used to avoid highly stressed components or excessive shrinkage. The thickness of the gate preferably should be equal to the part thickness. When using submarine gates, the minimum recommended diameter is 0.8 mm.

Hot runner molds may be used effectively, but precise temperature control is necessary to prevent material drooling or freezing off at the nozzle. When hot runners are used, the size of the gates may be smaller than in the case of cold runners.

D.5.2.5 Chemical and Physical Properties

PA 12 is a linear, semicrystalline-crystalline thermoplastic derived from butadiene. It has properties similar to PA 11, but its crystal structure is different. PA 12 is a good electrical insulator and its properties are not as sensitive to humidity as other polyamides. It has good resistance to shock and many chemicals. It is extensively modified with plasticisers and reinforcements. In comparison to PA 6 and PA 66, these materials have a lower melting point and density, and a much lower moisture regain. It is not resistant to strong oxidizing acids.

Viscosity is determined by water content, temperature, and residence time. This material flows easily. Shrinkage is of the order of 0.005 to 0.02 mm/mm (0.5 to 2%). This depends on the specific grade, wall thickness, and processing conditions.

D.5.2.6 Major Manufacturers

- Arkema (Rilsan)
- Degussa AG (Vestamid)
- PolyOne (Edgetek)

D.5.3 PA 6

D.5.3.1 Generic Class

Polyamide 6, Nylon 6, or Polycaprolactam

D.5.3.2 Typical Applications

Used in many structural applications because of its good mechanical strength and rigidity. It is used in bearings because of its good wear resistance.

D.5.3.3 Injection Molding Processing Conditions

Drying: Since PA 6 absorbs moisture readily, care should be taken to ensure its dryness prior to molding. If the material is supplied in watertight packaging, the containers should be kept closed. If the moisture content is >0.2%, drying in a hot air oven at 80°C (176°F) for 16 hours is recommended. If the material has been exposed to air for more than 8 hours, vacuum drying at 105°C (221°F) for more than 8 hours is recommended.

Melt Temperature: 230–280°C (446–536°F); 250–300°C (482–572°F) for reinforced grades.

Mold Temperature: 80–90°C (176–194°F). Mold temperature significantly influences the crystallinity level which in turn affects the mechanical properties. For structural parts, a high degree of crystallization is required and mold temperatures of 80–90°C (176–194°F) are recommended. High mold temperatures are also recommended for thin-walled parts with long flow lengths. Increasing the mold temperature increases the strength and hardness, but the toughness is decreased. When the wall thickness is greater than 3 mm, a cold mold is recommended (20–40°C / 68–104°F), which leads to a higher and more uniform degree of crystallinity. Glass reinforced resins are always processed at mold temperatures greater than 80°C (176°F).

Injection Pressure: Generally between 75 and 125 MPa (11,000 and 18,000 psi), depending on material and product design.

Injection Speed: High (slightly lower for reinforced grades).

D.5.3.4 Runners and gates

The gate location is important because of very fast freeze-off times. Any type of gate may be used; the aperture should not be less than half the thickness of the part. When hot runners are used, the size of the gates can be smaller than when cold runners are used because premature freeze-off is prevented. When using submarine gates, the minimum diameter of the gate should be 0.75 mm.

D.5.3.5 Chemical and Physical Properties

The molecular structure of polyamides consist of amide (CONH) groups joined by linear aliphatic sections (based on methylene groups). The toughness, rigidity, crystallinity, and thermal resistance of polyamide resins are because of the strong interchain attraction caused by the polarity of the amide groups. The CONH groups also cause a lot of moisture absorption.

Nylon 6 is produced by polymerization of caprolactam. The chemical and physical properties are similar to that of PA 66. However, its melting point is lower than PA 66 and it has a wider processing temperature range. Its impact strength and solvent resistance are better than PA 66, but its moisture absorption is higher. Many properties are affected by moisture absorption, which must be taken into account when designing with this resin. Various modifiers are added to improve mechanical properties; glass is one of the most commonly used fillers. The ddition of elastomers such as EPDM or SBR improves impact resistance.

For unfilled grades, shrinkage is of the order of 0.01 to 0.015 mm/mm (1 to 1.5%). The addition of glass fibers reduce the shrinkage to as low as 0.3% in the flow direction (but could be as high as 1% in the cross-flow direction). The post-molding shrinkage is affected mainly by the crystallinity level and moisture absorption. The actual shrinkage is a function of part design, wall thickness, and processing conditions.

D.5.3.6 Major Manufacturers

- BASF (Ultramid B)
- DuPont (Zytel)
- DSM (Akulon)

D.5.4 PA 66

D.5.4.1 Generic Class

Polyamide 66, Nylon 66, or Poly (hexamethylene adipamide)

D.5.4.2 Typical Applications

PA66 competes with PA 6 for most applications. PA 66 is heavily used in the following:

- The automotive industry
- Appliance housings
- Where impact resistance and strength are required

D.5.4.3 Injection-molding Processing Conditions

Drying: Drying is not required if the material is sealed prior to molding; however, if the containers are left open, drying in a hot air oven at 85°C (185°F) is recommended. If the moisture content is > 0.2%, vacuum drying at 105°C (220°F) for 12 hours is recommended.

Melt Temperature: 260–290°C (500–554°F); 275–280°C (527–536°F) for glass filled grades. Melt temperatures above 300°C (572°F) should be avoided.

Mold Temperature: 80°C (176°F) suggested. Mold temperature affects crystallinity level, which in turn affects physical properties. In the case of thin-walled parts, crystallinity changes with time if mold temperatures of less than 40°C (104°F) are used. In such cases, annealing may be needed to retain dimensional stability.

Injection Pressure: Generally between 75 and 125 MPa (11,000 and 18,000 psi), depending on material and product design.

Injection Speed: High (slightly lower for reinforced grades).

D.5.4.4 Runners and Gates

The gate location is important because of very fast freeze-off times. Any type of gate may be used; the aperture should not be less than half the thickness of the part. When hot runners are used, the size of the gates can be smaller than when cold runners are used, because premature freeze-off is prevented. When using submarine gates, the minimum diameter of the gate should be 0.75 mm.

D.5.4.5 Chemical and Physical Properties

PA 66 homopolymer is produced by the polymerization of hexamethylene diamine and adipic acid (a dibasic acid). Among commercially available polyamides, PA 66 has one of the highest melting points. It is a semicrystalline-crystalline material. The resins have strength and stiffness that is retained at elevated temperatures. It does absorb moisture after molding, but the retention is not as much as in the case of PA 6. Moisture absorption depends on the composition of the material, wall thickness, and environmental conditions. Dimensional stability and properties are all affected by the amount of moisture absorption which must be taken into account for product design.

Various modifiers are added to improve mechanical properties; glass is one of the most commonly used fillers. The addition of elastomers such as EPDM or SBR improves impact resistance.

The viscosity is low and, therefore, it flows easily (but not as easily as PA 6). This allows molding of thin components. The viscosity is very sensitive to temperature. Shrinkage is of the order of 0.01 to 0.02 mm/mm (1 to 2%). The addition of reinforcing glass fibers reduces the shrinkage to 0.2 to 1%. Differential shrinkage in the flow and cross-flow directions is quite high. Mineral fillers yield more isotropic moldings. PA 66 is resistant to most solvents but not to strong acids or oxidizing agents.

D.5.4.6 Major Manufacturers

- BASF (Ultramid A)
- DSM (Akulon)
- DuPont (Zytel)
- Solutia (Vydyne)

D.5.5 PBT

D.5.5.1 Generic Class

Polybutylene Terephthalates

D.5.5.2 Typical Applications

- Household appliances (e.g., food processor blades, vacuum cleaner parts, fans, hair dryer housings, coffee makers)
- Electronics (e.g., switches, motor housings, fuse cases, key caps for computer keyboards, connectors, fiber-optic buffer tubing)
- Automotive (e.g., grilles, body panels, wheel covers, and components for doors and windows)

D.5.5.3 Injection-molding Processing Conditions

Drying: This material is sensitive to hydrolysis at high temperatures. It is therefore important to dry the material prior to molding. Suggested drying conditions (in air) are 120°C (248°F) for 6 to 8 hours [or 150°C (300°F) for 2 to 4 hours]. Moisture levels must be below 0.03%. When using a desiccant dryer, drying at 120°C (248°F) for 2.5 hours is recommended.

Melt Temperature: 220–280°C (428–536°F); aim: 250°C (482°F).

Mold Temperature: 40–60°C (104–140°F) for unreinforced grades. For other grades, a wide range of temperatures can be used, depending on the grade (15–80°C / 59–176°F). Cooling channels should be properly designed to minimize part warpage. The heat removal must be fast and uniform. Cooling channels of 12 mm diameter are recommended.

Injection Pressure: Moderate (up to maximum of 150 MPa / 21750 psi).

Injection Speed: Fastest possible speeds should be used (due to fast solidification of PBTs).

D.5.5.4 Runners and Gates

Full round runners are recommended to impart maximum pressure transmission (rule of thumb: runner diameter = part thickness + 1.5 mm). A wide variety of gates may be used. Hot runners may also be used, taking care to avoid drool and material degradation. Gate diameters or depths should preferably be between 0.8 to 1.0 times the part thickness. When using submarine gates, the minimum recommended diameter is 0.75 mm.

D.5.5.5 Chemical and Physical Properties

PBT is one of the toughest engineering thermoplastics. It is a semicrystalline resin and has excellent chemical resistance, mechanical strength, electrical properties (high dielectric strength and insulation resistance), and heat resistance, all of which are stable over a broad range of environmental conditions. It has very low moisture absorption.

PBT, which is a polyester, is produced by the polycondensation reaction of dimethyl terephthalate an butanediol.

Tensile strength ranges from 50 MPa (7,250 psi) for unfilled grades to 170 MPa (24,650 psi) for glass-reinforced grades. High levels of glass fillers make the material more brittle. Crystallization is rapid, which could cause warpage from non-uniform cooling. In the case of glass-filled grades, shrinkage is reduced in the flow direction, but in the cross-flow direction it may be equal to that of the base resin. Shrinkage is on the order of 0.015 to 0.028 mm/mm (1.5 to 2.8%). A 30% glass-filled resin has a shrinkage range of 0.3 to 1.6%. The melting point (approximately 225°C/437°F) and heat distortion temperatures are lower than that of PET. The Vicat softening point is approximately 170°C (338°F). The glass transition temperature ranges from 22 to 43°C (71 to 109°F).

The melt viscosity is fairly low and due to fast crystallization rates, cycle times are typically low.

D.5.5.6 Major Manufacturers

- BASF (Ultradur)
- LANXESS (Pocan)
- GE Plastics (Valox)
- Ticona (Celanex)

D.5.6 PC

D.5.6.1 Generic Class

Polycarbonate

D.5.6.2 Typical Applications

- Electronic and business equipment (e.g., computer parts, connectors)
- Appliances (e.g., food processors, refrigerator drawers)
- Transportation (e.g., headlights, taillights, instrument panels)

D.5.6.3 Injection-molding Processing Conditions

Drying: PC resins are hygroscopic and pre-drying is important. Recommended drying conditions are 100 to 120°C (212 to 248°F) for 3 to 4 hours. Moisture content must be less than 0.02% prior to processing.

Melt Temperature: 260–340°C (500–644°F); higher range for low MFR resins and vice-versa.

Mold Temperature: 70–120°C (158–248°F); higher range for low MFR resins and vice-versa.

Injection Pressure: As high as possible for rapid molding.

Injection Speed: Slow injection speeds when small or edge gates are used; high speeds for other types of gates.

D.5.6.4 Chemical and Physical Properties

Polycarbonate is a polyester of carbonic acid. All general-purpose polycarbonates are based on bisphenol A. The bisphenol A component of the molecule contributes to the high glass transition temperature (150°C/302°F). The rotational mobility of the carbonyl group within the molecule contributes to the high ductility and toughness of the resin.

PC is an amorphous engineering resin with exceptionally good impact strength, heat resistance, clarity, sterilizability, flame retardancy, and stain resistance. The notched Izod impact strength of PC is very high and mold shrinkage is low and consistent (0.1 to 0.2 mm/mm).

High-molecular-weight PCs (which translates to low melt flow rate) have higher mechanical properties, but processability of such resins becomes difficult. The type of PC chosen for a particular application should be based on the desired criteria (for high impact properties, use a low-MFR PC; conversely, for optimal processability, use a high-MFR PC).

The melt viscosities are typically Newtonian up to shear rates of 1000 1/s and decrease beyond that. The Heat Deflection Temperature Under Load is typically between 130 and 140°C (266 and 284°F) and the Vicat Softening Point is typically around 155°C (311°F).

D.5.6.5 Major Manufacturers

- Bayer (Apec, Makrolon)
- Dow Chemical (Calibre)
- DSM (Xantar)
- GE Plastics (Lexan)
- LNP (Thermocomp)
- Teijin Chemical (Panlite)

D.5.7 PC / ABS

D.5.7.1 Generic Class

Polycarbonate | Acrylonitrile-Butadiene-Styrene Blend

D.5.7.2 Typical Applications

- Computer and business machine housings
- Electrical applications
- Cellular phones
- Lawn and garden equipment
- Automotive components (instrument panels, interior trim, and wheel covers)

D.5.7.3 Injection-molding Processing Conditions

Drying: Drying is required prior to processing. Moisture content should be less than 0.04% to ensure stable processing parameters. Drying at 90 to 110°C (194 to 230°F) for 2 to 4 hours is recommended.

Melt Temperature: 230–300°C (446–572°F).

Mold Temperature: 50–100°C (122–212°F).

Mold Temperature: Part dependent.

Injection Pressure: Part dependent.

Injection Speed: As high as possible.

D.5.7.4 Chemical and Physical Properties

PC/ABS offers combined properties of PC and ABS (high processability of ABS along with excellent mechanical properties and impact and heat resistance of PC). The ratio of the two components affects the heat resistance. The blend exhibits excellent flow characteristics.

D.5.7.5 Major Manufacturers

- Bayer (Bayblend)
- Dow Chemicals (Pulse)
- DSM (Xantar)
- GE Plastics (Cycoloy)
- Teijin Chemical (Multilon)

D.5.8 PC/PBT

D.5.8.1 Generic Class

Polycarbonate | Polybutyleneterephthalate Blend

D.5.8.2 Typical Applications

- Gear cases and automotive (bumpers)
- Applications that require chemical and corrosion resistance, high heat resistance, high impact strength over wide temperature ranges, and high dimensional stability

D.5.8.3 Injection-molding Processing Conditions

Drying: 110–135°C (230–275°F) for approximately 4 hours.

Melt Temperature: 230–280°C (402–536°F) depending on specific grade.

Mold Temperature: 40–85°C (104–185°F).

D.5.8.4 Chemical and Physical Properties

PC|PBT blends offer a combination of properties of PC and PBT: high toughness and dimensional stability of PC and good chemical and heat resistance and lubricity of crystalline PBT.

D.5.8.5 Major Manufacturers

- MRC Polymers (Naxaloy)
- GE Plastics (Xenoy)

D.5.9 HDPE

D.5.9.1 Generic Class

High Density Polyethylene

D.5.9.2 Typical Applications

Major use is in blow-molding (packaging) applications such as:

- Containers in refrigeration units
- Storage vessels
- Household goods (kitchenware)
- Seal caps
- Bases for PET bottles

D.5.9.3 Injection-molding Processing Conditions

Drying: Not normally necessary if stored properly.

Melt Temperature: 180–280°C (356–536°F). For high-molecular-weight resins, the suggested melt temperature range is 200 to 250°C (392 to 482°F).

Mold Temperature: 20–95°C (68–194°F), with higher temperatures for wall thickness of up to 6 mm and lower temperatures for wall thicknesses greater than 6 mm.

Injection Pressure: 70–105 MPa (10,000–15,000 psi)

Injection Speed: Fast injection speeds are recommended; profiled speeds reduce warpage in the case of components with a large surface area.

D.5.9.4 Runners and Gates

Diameters of runners range from 4 to 7.5 nun (typically 6 mm). Runner lengths should be as short as possible. All types of gates may be used. Gate lands should not exceed 0.75 mm in length. Ideally suited for hot runner molds; an insulated hot tip runner is preferred when there are frequent color changes.

D.5.9.5 Chemical and Physical Properties

High-density polyethylene is produced from polymerization of ethylene (lower temperature and pressure conditions are used compared to the production of low-density polyethylene). The material is free from branching, which is made possible by the use of stereospecific catalysts. Because of molecular regularity, HDPE has a high level of crystallinity (compared to LDPE).

Higher levels of crystallinity contribute to higher density, tensile strength, heat distortion temperature, viscosity, and chemical resistance. HDPE is more resistant to permeability than LDPE. The impact strength is lower. The properties of HDPE are controlled by the density, and molecular weight distributions. Injection molding grades typically have a narrow molecular weight distribution.

When the density is 0.91 to 0.925 g/cm3, the material is known as Type 1; Type 2 materials have densities in the range of 0.926 to 0.94 g/cm3; and Type 3 materials have densities in the range of 0.94 to 0.965 g/cm3.

The material flows easily and the MFR ranges from 0.1 to 28. Higher molecular weights (lower MFR grades) have better impact resistance. Being a semicrystalline material, the molding shrinkage is high (order of 0.015 to 0.04 mm/mm or 1.5 to 4%). This depends on the degree of orientation and level of crystallinity in the part (which in turn depend on processing conditions and part design).

PE is susceptible to environmental stress cracking, which can be minimized by reducing internal stresses by proper design and using the lowest MFR material at a particular density level. HDPE is soluble in hydrocarbons at temperatures greater than 60°C, but resistance to these materials is greater than that for LDPE.

D.5.9.6 Major Manufacturers

- ExxonMobil (Escorene, Paxon)
- Basell (Lupolen)
- Dow (Dowlex)
- Arkema (Lacqtene)
- Equistar (Alathon)
- Chevron Phillips (Marlex)
- Nova Chemicals (Sclair)

D.5.10 LDPE

D.5.10.1 Generic Class

Low Density Polyethylene

D.5.10.2 Typical Applications

- Closures
- Bowls
- Bins
- Pipe couplings

D.5.10.3 Injection Molding Processing Conditions

Drying: Not usually necessary.

Melt Temperature: 180–280°C (355–535°F).

Mold Temperature: 20–70°C (68–158°F); for uniform and economic heat removal, it is recommended that the cooling channel diameters be at least 8 mm and the distance from the surface of the mold to the edge of the cooling channel be not more than 1.5 times the diameter of the cooling channel.

Injection Pressure: Up to 150 MPa (21,750 psi).

Injection Speed: Fast speeds are recommended; profiled speeds can limit warpage problems of large surface area parts.

D.5.10.4 Runners and Gates

All conventional types may be used; LDPE is well-suited for hot runner molds. Insulated hot tip runners are preferred for frequent color changes.

D.5.10.5 Chemical and Physical Properties

Low density polyethylene is produced by the polymerization of ethylene at high pressure and temperature. The resin is semicrystalline-crystalline. The crystallinity level is low because of chain branching. The material is tough but possesses moderate tensile properties and exhibits creep. However, it has good impact and chemical resistance. It is an easy flow material because of long chain branching.

Commercial materials have densities in the range of 0.91 to 0.94 g/cm3. LDPE is permeable to gases and vapors. Very close tolerances are not possible with this resin and its relatively large coefficient of thermal expansion makes it less suitable for long term applications.

Shrinkage is of the order of 0.02 to 0.05 mm/mm (2 to 5%) when density is between 0.91 to 0.925 g/cm3. When density is between 0.926 to 0.940 g/cm3, the shrinkage is of the order of 1.5 to 4%. Actual shrinkage values depend on the molding conditions.

LDPE is resistant to many solvents at room temperatures but aromatic and chlorinated hydrocarbons cause swelling. Like HDPE, it is also susceptible to environmental stress cracking.

D.5.10.6 Major Manufacturers

- Arkema (Lacqtene)
- Basell (Lupolen)
- Chevron Phillips
- Dow (Dowlex LDPE)
- Nova Chemicals (Novapol)
- Quantum Chemicals

D.5.11 PEI

D.5.11.1 Generic Class

Polyetherimide

D.5.11.2 Typical Applications

- Automotive (engine components: temperature sensors, fuel and air handling devices)
- Electrical/electronics (connector materials, printed circuit boards, circuit chip carriers, explosion proof boxes)
- Packaging applications
- Aircraft (interior materials)
- Medical (surgical staplers, tool housings, nonimplant devices)

D.5.11.3 Injection-molding Processing Conditions

Drying: PEI absorbs moisture and can cause material degradation. Moisture content should be less than 0.02%. Suggested drying conditions are 150°C (302°F) for 4 hours in a desiccant dryer (6 hours for reinforced and blended grades).

Melt Temperature: 340–400°C (644–752°F) for unreinforced grades; 340–440°C (644–824°F) reinforced grades.

Mold Temperature: 70–175°C (108–347°F). Aim: 140°C.

Injection Pressure: 7–150 MPa (10,000–22,000 psi) typical.

Injection Speed: As high as possible.

D.5.11.4 Chemical and Physical Properties

PEIs are amorphous materials whose chemical structure consists of repeating aromatic imide and ether units. This accounts for its high temperature resistance. It also leads to high stiffness, and modifiers are used to make the material processable. PEIs are very stiff and strong even without reinforcements. They have excellent thermal stability making it possible to use them in high temperature applications. They have good flame and chemical resistance and good electrical insulation properties. The glass transition temperature is high (215°C/ 419°F). It exhibits low shrinkage and highly isotropic mechanical properties.

D.5.11.5 Major Manufacturers

- GE Plastics (Ultem)
- RTP Company

D.5.12 PET

D.5.12.1 Generic Class

Polyethylene Terephthalate

D.5.12.2 Typical applications

- Automotive (structural components, such as mirror backs and grille supports; electrical parts, such as head lamp reflectors and alternator housings)
- Electrical applications (motor housings, electrical connectors, relays, and switches, microwave oven interiors)
- Industrial applications (furniture chair arms, pump housings, hand tools)

D.5.12.3 Injection Molding Processing Conditions

Drying: Drying is essential prior to molding. PETs are very sensitive to hydrolysis. Recommended drying conditions are 120 to 165°C (248 to 329°F) for 4 hours. The moisture content should be less than 0.02%.

Melt Temperature: 265–280°C (509–536°F) for unfilled grades; 275–290°C (527–554°F) for glass-reinforced grades.

Mold Temperature: 80–120°C (176–248°F). Preferred range: 100–110°C; (212–230°F).

Injection Pressure: 300–130 MPa (4,350–19,000 psi).

Injection Speed: High speeds without causing embrittlement.

D.5.12.4 Runners and Gates

All conventional types of gates may be used; gates should be 50 to 100% of the part thickness.

D.5.12.5 Chemical and Physical Properties

PET is an aromatic polyester produced from polymerization of either terephthalic acid (TPA) or dimethyl ester terephthalic acid (DMT) and ethylene glycol (EG). The glass transition is approximately 165°C (330°F) and the resin crystallizes over a temperature range from 120 to 220°C (248 to 428°F).

PET is highly sensitive to moisture at high temperatures and exhibits excessive warpage when reinforced with glass fibers. Promotion of crystallinity is achieved through adding nucleating agents and crystal growth accelerators. Crystalline moldings exhibit high modulus, gloss, and heat distortion temperatures. Warpage is minimized by addition of particulate fillers such as mica. When low mold temperatures are used, clear moldings can be obtained with unfilled PETs.

D.5.12.6 Major Manufacturers

- BASF (Petra)
- DuPont (Rynite)
- Eastman Chemical (Eastapak)
- Ticona (Impet)

D.5.13 PETG

D.5.13.1 Generic Class

Glycol-modified PET; Copolyesters

D.5.13.2 Typical Applications

PETGs offer a desirable combination of properties such as clarity, toughness, and stiffness. Applications include:

- Medical devices (test tubes and bottles)
- Toys
- Displays
- Lighting fixtures
- Face shields
- Refrigerator crisper pans

D.5.13.3 Injection-molding Processing Conditions

Drying: Drying is essential for PETG prior to injection molding. The moisture level must be below 0.04%. Drying temperature is not to exceed 66°C (150°F). Drying at approximately 65°C (149°F) for 4 hours is recommended.

Melt Temperature: 220–290°C (428–554°F). The melt temperature is grade specific.

Mold Temperature: 10–30°C (50–86°F). Recommended: 15°C (60°F).

Injection Pressure: 30–130 MPa (4,350–19,000 psi)

Injection Speed: High speeds without causing embrittlement.

D.5.13.4 Chemical and Physical Properties

PETGs (or copolyesters) are glycol modified PETs; the modification is done by adding a second glycol during polymerization. The resulting molecular structure is irregular and the resin is clear and amorphous with a glass transition temperature of 88°C (190°F). PETGs can be processed over a wider processing range than conventional PETs and offer good combination of properties such as toughness, clarity, and stiffness.

D.5.13.5 Major Manufacturers

- Eastman (Eastar)
- Noveon (Stat-Rite)
- RTP Company (PermaStat)

D.5.14 PMMA

D.5.14.1 Generic Class

Polymethyl Methacrylate

D.5.14.2 Typical Applications

- Automotive (signal light devices, instrument panels)
- Medical (blood cuvettes)
- Industrial (video discs, lighting diffusers, display shelving)
- Consumer (drinking tumblers, stationery accessories)

D.5.14.3 Injection-molding Processing Conditions

Drying: PMMA is hygroscopic and must be dried prior to molding. Drying at 90°C (194°F) for 2 to 4 hours is recommended.

Melt Temperature: 240–280°C (460–536°F).

Mold Temperature: 35–80°C (90–176°F).

Injection Speed: Moderate.

D.5.14.4 Chemical and Physical Properties

Pellets for injection molding are made either by bulk polymerization of methyl methacrylate followed by extrusion and pelletization or by polymerization in an extruder. Formulations vary by molecular weight and physical properties such as flow rate, heat resistance, and toughness. Higher molecular weight grades are tougher than lower molecular weight grades. High flow formulations are generally preferred for molding.

Heat deflection temperature under load varies from 75°C (167°F) for high flow materials to 100°C (212°F) for low flow (high molecular weight) materials.

PMMA has excellent optical properties and weatherability. The white light transmittance is as high as 92%. Molded parts can have very low birefringence, which makes it ideally suited as a material for video discs.

PMMA exhibits room temperature creep. The initial tensile strength is high but under long term, high stress loading, it exhibits stress craze. Impact strength is good but it does show some notch sensitivity.

D.5.14.5 Major Manufacturers

- Arkema/Atoglas (Plexiglas)
- BASF (Lucryl)
- Degussa (Plexiglas)
- Cyro Industries (Acrylite, Cyrolite)
- LG Chemical

D.5.15 POM

D.5.15.1 Generic Class

Polyacetal or Polyoxymethylene

D.5.15.2 Typical Applications

Acetals have a low coefficient of friction and good dimensional stability. This makes it ideal for use in gears and bearings. Because of its high temperature resistance, it is used in plumbing (valve and pump housings) and lawn equipment.

D.5.15.3 Injection Molding Processing Conditions

Drying: Not usually required but resin should be stored in a dry atmosphere.

Melt Temperature: 190–230°C (374–446°F) for homopolymer; 180–210°C (356–410°F) for copolymer.

Mold Temperature: 80–105°C (122–221°F). Higher mold temperatures are preferred for precision molding to reduce post-molding shrinkage.

Injection Pressure: 70–120 MPa (10,000–17,500 psi).

Injection Speed: Medium to high.

D.5.15.4 Runners and Gates

Any type of gate may be used. When using tunnel gates, the short type is preferred. Insulated, hot tip runners are preferred for homopolymers; both internally and externally heated hot runners may be used in the case of copolymers.

D.5.15.5 Chemical and Physical Properties

Acetals are tough, resilient materials and exhibit good creep resistance, dimensional stability, and impact resistance even at low temperatures. Acetal resins are either homopolymers or copolymers. Homopolymers have better tensile strength, fatigue resistance, and hardness but are difficult to process. Copolymers have better thermal stability, chemical resistance, and processability. Both homopolymers and copolymers are crystalline and have low moisture absorption.

Copolymers may be used continuously at air temperatures up to 100°C (212°F); homopolymers have slightly higher temperature resistance. Many grades of acetal resins are available, tailored to different applications.

High crystallinity levels of acetals lead to relatively high shrinkage levels of 0.02 to 0.035 mm/mm. Differential shrinkage is observed with reinforced grades.

D.5.15.6 Major Manufacturers

- BASF (Ultraform copolymers)
- DuPont (Delrin homopolymers)
- Ticona (Hostaform, Celcon copolymers)

D.5.16 PP

D.5.16.1 Generic Class

Polypropylene

D.5.16.2 Typical Applications

- Automotive (mostly mineral-filled PP is used: dashboard components, duct work, fans, and some under-hood components)
- Appliances (door liners for dishwashers, duct work for dryers, wash racks and lids for clothes washers, refrigerator liners)
- Consumer products (lawn/garden furniture, components of lawn mowers, sprinklers)

D.5.16.3 Injection-molding Processing Conditions

Drying: Not normally necessary if proper storage is used.

Melt Temperature: 220–280°C (428–536°F) not to exceed 280°C.

Mold Temperature: 20–80°C (68–176°F). Suggested: 50°C (122°F). The crystallinity level is determined by the mold temperature.

Injection Pressure: Up to 180 MPa (26,000 psi).

Injection Speed: Typically, fast injection speeds are used to minimize internal stresses; if surface defects occur, slow speed molding at a higher temperature is preferred. Machines capable of providing profiled speed are highly recommended.

D.5.16.4 Runners and Gates

In the case of cold runners, typical diameters range from 4 to 7 mm. Fully round sprues and runners are recommended. All types of gates can be used. Typical pin gate diameters range from 1 to 1.5 mm, but diameters as low as 0.7 mm may be used. In case of edge gating, the minimum gate depth should be half the wall thickness and the width should be at least double the thickness. Hot runners can readily be used for molding PP.

D.5.16.5 Chemical and Physical Properties

PP is produced by the polymerization of propylene using stereospecific catalysts. Mainly, isotactic PP is produced (the methyl groups lie on one side of the carbon chain). This linear plastic is semicrystalline because of ordered molecular structure. It is stiffer than PE and has a higher melting point. The PP homopolymer becomes very brittle at temperatures higher than 0°C (32°F) and for this reason, many commercially available grades are random copolymers with 1 to 4% ethylene or block copolymers with higher ethylene content. Copolymers have a lower heat distortion temperature (approximately 100°C/212°F), less clarity, gloss, and rigidity, but greater impact strength. The material becomes tougher as the ratio of ethylene increases. The Vicat softening point is approximately 150°C (302°F). Because of high levels of crystallinity, the surface hardness and scratch resistance is higher for these materials.

PP does not have environmental stress cracking problems. PP is usually modified by addition of glass fibers, mineral fillers, or thermoplastic rubbers. The MFR of PP ranges from 1 to 40; lower MFR materials have better impact strength but lower tensile strength. The copolymer is tougher than the homopolymer of the same MFR. The viscosity is more shear and temperature sensitive than PE.

Due to crystallinity, the shrinkage is relatively high (order of 0.018 to 0.025 mm/mm or 1.8 to 2.5%). The shrinkage is more uniform than PE-HD (the difference in flow and cross-flow shrinkage is typically less than 0.2%). Addition of 30% glass reduces the shrinkage to approximately 0.7%.

Both homopolymer and copolymer PP offer excellent resistance to moisture and good chemical resistance to acids, alkalis, and solvents. However, it is not resistant to aromatic hydrocarbons such as benzene, and chlorinated hydrocarbons such as carbon tetrachloride. It is not as resistant to oxidation at high temperatures as PE.

D.5.16.6 Major Manufacturers

- Albis
- Amoco
- Arkema (Appryl)
- A. Schulman (PolyFort)
- BASF (Novolen)
- Borealis
- Ebbtide Polymers (Aclo Accutech)
- Exxon Chemical
- Montel
- Phillips Sumika (Marlex)

D.5.17 PPE / PPO

D.5.17.1 Generic Class

Polypropylene Ether Blends

D.5.17.2 Typical Applications

- Household appliances (dishwashers, washing machines)
- Electrical applications, such as control housings, fiber-optic connectors

D.5.17.3 Injection Molding Processing Conditions

Drying: Recommend drying before molding for approximately 2 to 4 hours at 100°C (212°F). PPOs have low levels of moisture absorption and can typically be molded as received.

Melt Temperature: 240–320°C (464–608°F); higher ranges for resins with higher levels of PPO.

Mold Temperature: 60–105°C (140–220°F).

Injection Pressure: 60–150 MPa (8,700–21,750 psi).

D.5.17.4 Runners and Gates

All gates can be used; tab and fan gates are preferred.

D.5.17.5 Chemical and Physical Properties

PPO is poly (2,6 dimethyl p-phenylene) oxide. The ether linkages offer easier processability. Copolymers are referred to as PPEs (Polyphenylene Ethers). Typically, the commercially available PPOs (or PPEs) are blended with other thermoplastic materials such as PS (or HIPS), Nylon, etc. These blends are still referred to as PPOs or PPEs.

The blends offer superior processability compared to pure PPOs. Their viscosities are lower. A range of properties can be obtained depending on the ratios of PPO and PS. Blends with nylons (PA 6/6) offer improved chemical resistance and perform well at high temperatures. The water absorption is low and the molded products have excellent dimensional stability.

Blends with PS are amorphous whereas blends with Nylons are crystalline. The addition of glass fibers reduces shrinkage levels to 0.2%. These materials have excellent dielectric properties and a low coefficient of thermal expansion. The viscosity level depends on the ratio of the components in the blend; higher PPO levels increase the viscosity.

D.5.17.6 Major Manufacturers

- Custom Resins (Norpex)
- Degussa (Vestoran)
- GE Plastics (Noryl)

D.5.18 PS

D.5.18.1 Generic Class

Polystyrene

D.5.18.2 Typical Applications

- Packaging
- Housewares (tableware, trays)
- Electrical (transparent housings, light diffusers, insulating film)

D.5.18.3 Injection-molding Processing Conditions

Drying: Not usually required unless stored improperly. If drying is needed, the recommended conditions are 2 to 3 hours at 80°C (176°F).

Melt Temperature: 180–280°C (356–536°F). The upper limit is 250°C (482°F) for flame retardant grades.

Mold Temperature: Suggested: 20–70°C (68–158°F).

Injection Pressure: 20–60 MPa (3,000–8,700 psi).

Injection Speed: Fast speeds are recommended.

D.5.18.4 Runners and Gates

All types of conventional gates may be used.

D.5.18.5 Chemical and Physical Properties

General-purpose PS is produced by the polymerization of styrene. Most commercial grades are clear, amorphous polymers. PS offers excellent dimensional and thermal stability, optical clarity, and very little tendency to absorb moisture. It has good dielectric properties. It is resistant to water and dilute inorganic acids, but is attacked by strong oxidizing acids such as concentrated sulfuric acid, and is swollen by some organic solvents.

Processing shrinkage is typically between 0.4 and 0.7%.

D.5.18.6 Major Manufacturers

- Albis
- Arkema (Lacqrene)
- BASF (Polystyrol)
- Chevron
- Dow (Questra)
- Creanova (Vestyron)

D.5.19 PVC

D.5.19.1 Generic Class

Polyvinyl Chloride

D.5.19.2 Typical Applications

- Water distribution piping
- Home plumbing
- House siding
- Business-machine housings
- Electronics packaging
- Medical apparatus
- Packaging for foodstuffs

D.5.19.3 Injection-molding Processing Conditions

Drying: Not usually necessary, as PVC absorbs very little water.

Melt Temperature: 160-220°C (320-428°F).

Mold Temperature: 20–70°C (68–158°F).

Injection Pressure: Up to 150 MPa (21,750 psi).

Packing Pressure: Up to 100 MPa (14,500 psi).

Injection Speed: Relatively slow, to avoid material degradation.

D.5.19.4 Runners and Gates

All conventional gate types may be used; pin-point and submarine gates are used for molding small components and fan gates are typically used for thick sections. The minimum diameter of pin-point or submarine gates should be 1 mm and the thickness of fan gates should not be less than 1 mm.

Sprues should be as short as possible; typical runner sizes are 6 to 10 mm and should have a full round cross-section. Insulated hot runners and certain types of hot sprue bushings may be used with PVC.

D.5.19.5 Chemical and Physical Properties

Rigid (unplasticized) PVC is one of the most widely used plastic materials. It is produced from sodium chloride and natural gas. The repeat chemical structure is vinyl chloride. Additives are mixed with PVC to make it processable. PVC resins produced by suspension or mass polymerization techniques are the major types used for melt processing. PVC is substantially an amorphous material.

Some of the additives used include stabilizers, lubricants, processing aids, pigments, impact modifiers, and fillers. Some features of PVC include low combustibility, toughness (designed to be virtually unbreakable), good weatherability (including good color and impact retention, and no loss in stiffness), and excellent dimensional stability. PVC is highly resistant to oxidizing and reducing agents, and strong acids. However, unplasticized PVC is not recommended for environmental and continuous use above 60°C (140°F). It is not resistant to concentrated oxidizing acids such as sulfuric or nitric acid and is unsuitable for use with aromatic and chlorinated hydrocarbons.

It is very important to process the material at the correct melt temperature; otherwise severe problems from decomposition (which produces hydrochloric acid which in turn accelerates decomposition) could result.

PVC is a relatively stiff flow material and has a narrow processing range. The molecular weight determines the flow characteristics. Higher molecular weight materials are more difficult to process (this could be modified by addition of lubricants). Typically, however, relatively low molecular weight resins are used in molding). Shrinkage is fairly low (0.002 to 0.006 mm/mm or 0.2 to 0.6%).

D.5.19.6 Major Manufacturers

- Arkema (Lacovyl)
- BASF (Vinidur, Vinoflex)
- Creanova (Vestolit)
- Keysor-Century
- PolyOne (Geon)
- Teknor APEX

D.5.20 SAN

D.5.20.1 Generic Class

Styrene Acrylonitrile

D.5.20.2 Typical Applications

- Electrical (receptacles, mixer bowls, housings, etc. for kitchen appliances; refrigerator fittings;chassis for television sets; cassette boxes)
- Automotive (head lamp bodies, reflectors, glove compartments, instrument panel covers)
- Household appliances (tableware, cutlery, beakers)
- Cosmetic packs

D.5.20.3 Injection Molding Processing Conditions

Drying: Under improper storage conditions, SAN absorbs moisture; it is recommended that it be dried at 80°C (176°F) for 2 to 4 hours prior to molding.

Melt Temperature: 200–270°C (392–518°F). For most applications, 230–260°C (446–500°F); the lower end of the range is used for molding thick-walled components.

Mold Temperature: 40–80°C (104–176°F). SAN solidifies rapidly at higher temperatures; in case of reinforced grades, the mold temperatures should not be less than 60°C (140°F). The cooling system must be well designed because the mold temperature affects the parts appearance and shrinkage and warpage.

Injection Pressure: 35–130 MPa (5,000–20,000 psi).

Injection Speed: High speeds are recommended.

D.5.20.4 Gates

All conventional gate types may be used. The gates must be of proper size to aid in processing without causing streaks, bum marks, or voids.

D.5.20.5 Chemical and Physical Properties

SAN copolymers are produced by the polymerization reaction of styrene and acrylonitrile. They are strong, transparent materials. The styrene component imparts clarity, stiffness, and processability and the acrylonitrile component imparts chemical and thermal resistance.

They have excellent load bearing capacity and rigidity; good resistance to chemicals, heat deformation, and cyclic temperature loads; and dimensional stability. The properties depend on the acrylonitrile content and commercial grades offer different acrylonitrile molecular masses. The addition of glass fibers enhances rigidity and resistance to heat deformation, and decreases the coefficient of linear thermal expansion.

The Vicat softening point is approximately 110°C (230°F) and the deflection temperature under load is approximately 100°C (212°F).

Shrinkage ranges from 0.003 to 0.007 mm/mm (0.3 to 0.7%).

D.5.20.6 Major Manufacturers

- BASF (Luran)
- Dow Chemical (Tyril)
- LANXESS (Lustran)
- Network Polymers

D.5.21 Additives, Fillers, and Reinforcements

Additives, fillers, and reinforcements are used to change and improve the physical and mechanical properties of plastics. In general, reinforcing fibers increase the mechanical properties of polymer composites while particular fillers of various types increase the modulus.

Table 3 lists a variety of additives, fillers, and reinforcements and their effects on the polymer properties.

Table 3: Effects of additives, fillers, and reinforcements on polymer properties

Additive/Filler/ Reinforcement	Common Materials	Effects on Polymer Properties
Reinforcing fibers	Boron Carbon Fibrous minerals Glass Kevlar	Increases tensile strength. Increases flexural modulus Increased heat-deflection temperature (HDT) Resists shrinkage and warpage
Conductive fillers	Aluminum powders Carbon fiber Graphite	Improves electrical and thermal conductivity
Coupling agents	Silanes Titanates	Improves interface bonding between polymer matrix and fibers
Flame retardants	Chlorine Bromine Phosphorours Metalic salts	Reduced the occurrence and spread of combustion
Extender fillers	Calcium carbonate Silica Clay	Reduces material cost
Plasticizers	Monomeric liquids Low-molecular weight materials	Improves melt flow properties Enhances flexibility
Colorants (pigments and dyes)	Metal oxides Chromates Carbon blacks	Provides color fastness Protects from thermal and UV degradation (with carbon blacks)
Blowing agents	Gas Azo compounds Hydrazine derivatives	Generates a cellular form to obtain a low-density material

D.5.22 Modifying Polymer Properties

Electrical properties can be affected by many fillers. For example, by adding conductive fillers an electromagnetic shielding property can be built into plastics, which are normally poor electrical conductors. Antistatic agents can be used to attract moisture, reducing the build-up of static charge.

Coupling agents are added to improve the bonding of the plastic matrix and the reinforcing fibers. Different fillers are used to lower the cost of materials. Other additives include flame retardants to reduce the likelihood of combustion, lubricants to reduce the viscosity of the molten plastic, plasticizers to increase the flexibility of the materials, and colorants to provide colorfastness.

D.5.23 Low-aspect Fillers

Fillers modify the properties and molding of the compound to which they are added. If the fillers are characterized with a low aspect ratio between the longest and the shortest dimensions, the basic properties will be less changed from those of the unfilled polymer. Fillers benefit plastics parts in the following ways:

- Shrinkage will be less.
- Thermal resistance may be improved.
- Strength, especially compressive strength, will be improved.
- Impact resistance will often be lower than for the unfilled polymer.
- Solvent resistance will often be improved.

D.5.24 High-aspect Fillers: Fibers

When the aspect ratio between the longest and the shortest dimension of the filler is large (for example, greater than 25) the filler can be characterized as a fiber. Fiber reinforcements will significantly affect the properties of the compounds to which they are added.

D.5.24.1 Fibers Impact Strength

Assuming good bonding between the fiber and the polymer matrix, the strength in the fiber direction will be significantly increased. If many fibers are oriented in the same direction, large differences will be noted between the modulus in the orientation direction and in the direction perpendicular to the orientation. The latter will be very close to that for the unfilled polymer.

D.5.24.2 Fibers Affect Shrinkage

The fibers will also have a significant effect on the shrinkage properties of the compound: shrinkage in the orientation direction will be much less than the shrinkage in the cross direction.

D.5.24.3 Importance of Predicting Fiber Orientation

Because the fiber orientation varies with the flow direction, in the thickness direction, and at weld line locations, it is important to be able to predict these orientations to determine the properties of the molded article.

Index