

Table of Contents

Part 1: Designing for Moldability

Chapter 1

Injection pressure

Equations

Factors that influence injection pressure requirements

Chapter 2

Filling pattern

Melt front velocity influences filling pattern

Chapter 3

MFA and MFV

Varying MFV

Equation

How flow dynamics affect orientation

Flow balance

Chapter 4

Material properties for part design

Stress-strain behavior

Fatigue

Impact strength

Thermal mechanical behavior

Design for strength

Short-term loading

Long-term loading

Repeated loading

High velocity and impact loading

Loading at extreme temperatures

Part thickness

Cycle time increases with thickness

Thick parts tend to warp

Thin, uniform parts improve surface quality

Reducing part thickness

Boosting structural integrity with ribs

Typical uses for ribs

Designing ribs

Design for assembly

Tolerances: fit between parts

Annular snap-fit joints

Torsion snap-fit joints

Fasteners

Inserts

Welding processes

Chapter 5

Runner systems

Runner size considerations

Payoffs of good runner design

Determining the number of cavities

Formulas

Planning the runner system layout

Balanced vs. unbalanced layouts

Determining sprue dimensions

Designing runner cross sections

Recommended cross-sectional designs

Hydraulic diameter and flow resistance

Determining runner dimensions

Example: using empirical data to calculate runner dimensions

Typical runner diameters

Hot runner systems

Types of hot runner systems

Runner balancing

Reducing runner diameter

Using tighter process controls

Design rules

Runner size

Raising melt temperature

Branched runners

Runner intersections

Hot runners

Easy ejection

Chapter 6

Gate design overview

Single vs. multiple gates

Gate dimensions

Gate location

Gate types

Manually trimmed gates

Automatically trimmed gates

Design rules

Gate location

Gate length

Gate size

Gate thickness

Freeze-off time

Fiber-filled materials

Chapter 7

Design Example: Optimizing gates and ram speed profile

C-MOLD Filling EZ analysis

C-MOLD Filling and Post Filling analysis

Part 2: Designing for Productivity and Performance

Chapter 8

Mold cooling system overview

Mold cooling system components

Cooling-channel configuration

Alternative cooling devices

Baffles

Bubblers

Thermal pins

Cooling slender cores

Cooling large cores

Cooling cylinder cores

Cooling system equations

Cooling time

Reynolds number and coolant flow

Chapter 9

Residual stress

Flow-induced residual stress

Thermal-induced residual stress

Process-induced vs. in-cavity residual stress

Chapter 10

Shrinkage and warpage

Shrinkage

Warpage

Warpage due to differential shrinkage

Design rules for shrinkage and warpage

Wall thickness

Balanced filling

Packing pressure

Cooling system

Residual stress

Chapter 11

Design Example: Reducing shrinkage and warpage

Description

Part 3: Troubleshooting

Chapter 12

Air traps

Problems caused by air traps

Remedies

Chapter 13

Black specks/black streaks

Causes of black specks/black streaks

Remedies

Chapter 14

Brittleness

Causes of brittleness

Remedies

Chapter 15

Burn marks

Causes of burn marks

Remedies

Chapter 16

Delamination

Causes of delamination

Remedies

Chapter 17

Dimensional variation

Causes of dimensional variation

Remedies

Chapter 18

Discoloration

Causes of discoloration

Remedies

Chapter 19

Fish eyes

Causes of fish eyes

Remedies

Chapter 20

Flash

Causes of flash

Remedies

Chapter 21

Flow marks

Causes of flow marks

Remedies

Chapter 22

Hesitation

Problems caused by hesitation

Remedies

Chapter 23

Jetting

Effects of jetting

Remedies

Chapter 24

Ripples

Cause of ripples

Remedies

Chapter 25

Short shot

Causes of short shot

Remedies

Chapter 26

Silver streaks

Causes of silver streaks

Remedies

Chapter 27

Sink marks and voids

Causes of sink marks and voids

Remedies

Chapter 28

Weld lines and meld lines

Problems caused by weld lines

Strength of weld lines

Remedies

Part 3: Reference

Appendix A

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

Benefits of the gas-assist process

Typical applications

Injection-compression molding

Lamellar (microlayer) injection molding

Live-feed injection molding

Low-pressure injection molding

Push-pull injection molding

Reactive molding

Types of reactive materials

Processing

Design considerations

Structural foam injection molding

Thin-wall molding

Appendix B

Injection molding machine

Machine specification

Machine function

Auxiliary equipment

Machine components

Injection system

Mold system

Hydraulic system

Clamping system

Molded system

Machine operating sequence

Screw operation

Back pressure

Injection speed

Screw rotation speed

Cushion

Secondary operations

Assembly

Decoration

Other secondary operations

Appendix C

Setting process conditions

Setting machine process conditions

Appendix D

What are plastics?

Polymer alloys and blends

Polymer composites

Classification of plastics

Structures and properties of plastics

Thermoplastics

Market share distribution of thermoplastics

Structures and properties of thermoplastics

Amorphous polymers

(Semi-)crystalline polymers

Liquid crystal polymers

Thermosets

Cross-linking (reaction)

Processing thermosets

Additives, fillers, and reinforcements

Modifying polymer properties

Low-aspect fillers

High-aspect fillers: fibers

Resin data table

Appendix E

How does plastic flow?

Deformation

Viscoelastic behavior

Melt shear viscosity

Newtonian fluid vs. non-Newtonian fluid

Shear-thinning behavior

Shear rate distribution

Pressure-driven flow

Melt flow length

Injection pressure vs. fill time

Flow instability

Fountain flow

Injection pressure

Pressure drives the melt

Pressure is the driving force that overcomes the resistance of polymer melt (see [Pressure-driven flow](#)), pushing the polymer to fill and pack the mold cavity. 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 1](#) below.

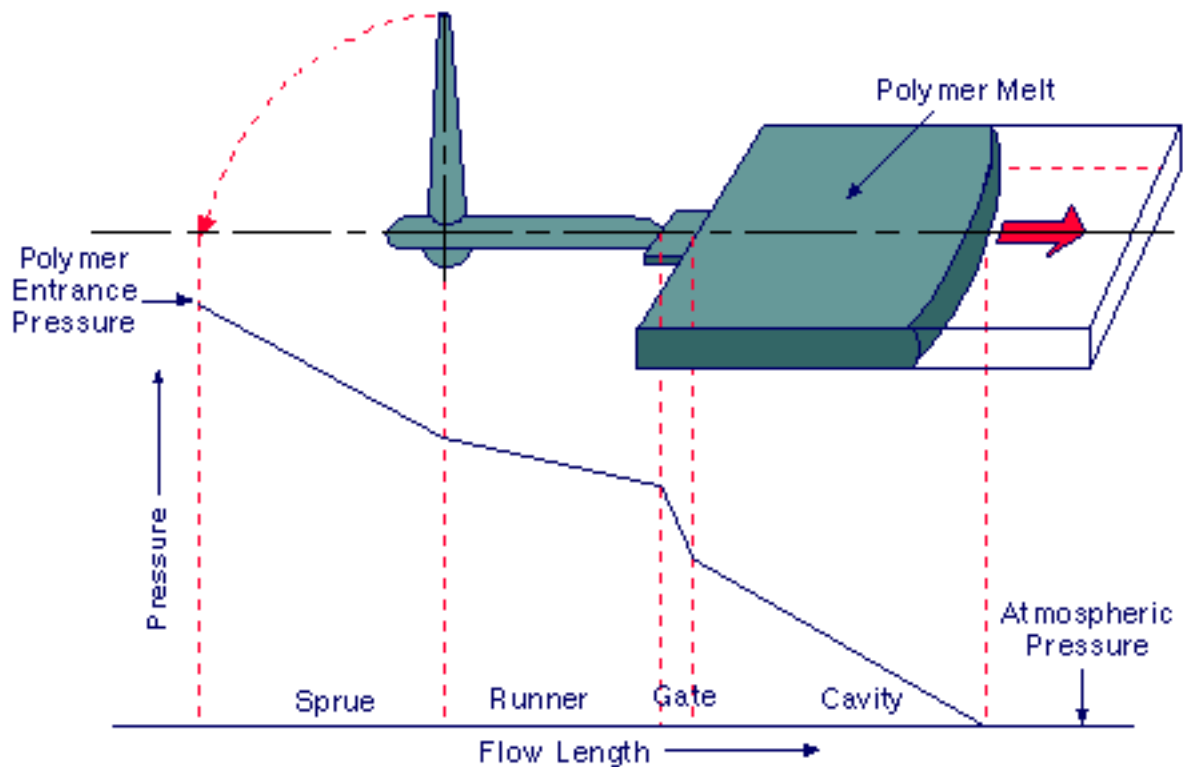


FIGURE 1. Pressure decreases along the delivery system and the cavity.

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](#) shows injection pressure as a function of several of these parameters.

► Circular channel flow

The melt flow in the sprue, runner, and cylindrical gates

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

► **Strip channel flow**

Such as melt flow in a thin cavity

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

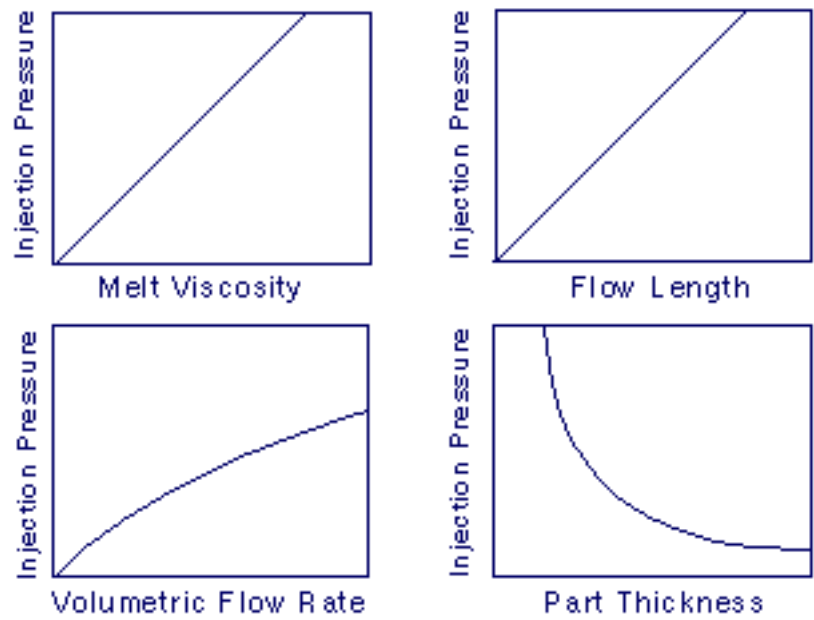
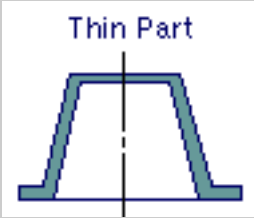
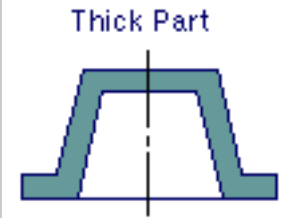

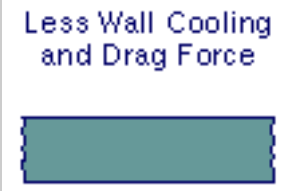
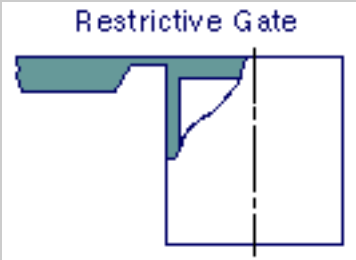
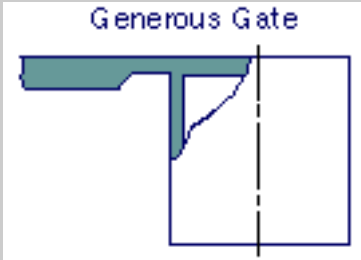
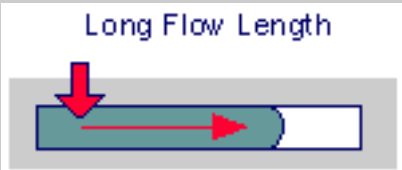
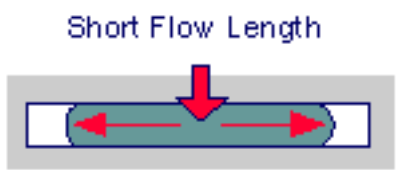
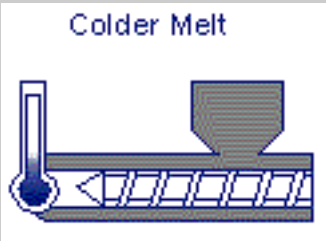
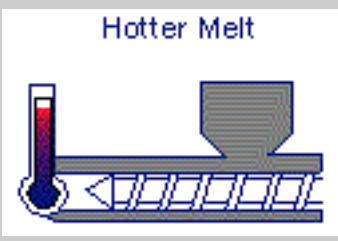
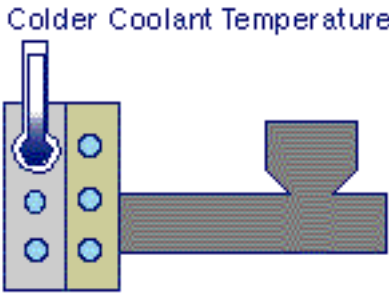
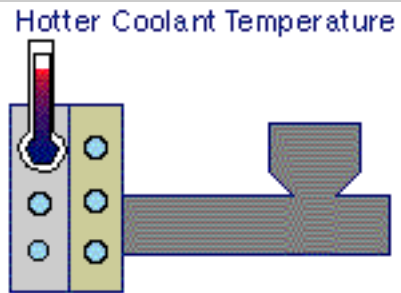


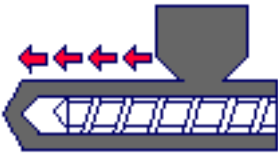
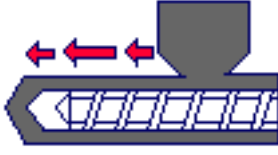
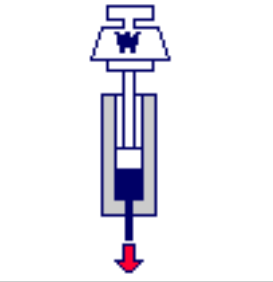
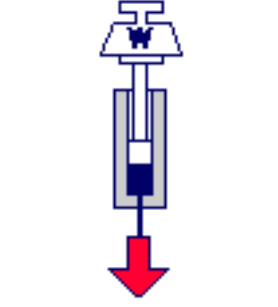
FIGURE 2. Injection pressure as a function of melt viscosity, flow length, volumetric flow rate, and part thickness

Factors that influence injection pressure requirements

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

Variable	Higher injection pressure required	Lower injection pressure required
PART DESIGN		

Part thickness	 <p>Thin Part</p> <p>A cross-sectional diagram of a trapezoidal part with a vertical dashed line through its center, indicating a thin profile.</p>	 <p>Thick Part</p> <p>A cross-sectional diagram of a trapezoidal part with a vertical dashed line through its center, indicating a thick profile.</p>
Part surface area	 <p>More Wall Cooling and Drag Force</p> <p>A diagram showing a rectangular part with a jagged, irregular top surface, representing increased surface area for cooling and drag.</p>	 <p>Less Wall Cooling and Drag Force</p> <p>A diagram showing a rectangular part with a smooth, flat top surface, representing reduced surface area for cooling and drag.</p>
GATE DESIGN		
Gate size	 <p>Restrictive Gate</p> <p>A diagram showing a gate with a narrow, curved cross-section, labeled as restrictive.</p>	 <p>Generous Gate</p> <p>A diagram showing a gate with a wide, straight cross-section, labeled as generous.</p>
Flow length	 <p>Long Flow Length</p> <p>A diagram showing a long, narrow channel with a red arrow indicating the flow direction from left to right.</p>	 <p>Short Flow Length</p> <p>A diagram showing a short, wide channel with a red arrow indicating the flow direction from left to right.</p>
PROCESSING CONDITIONS		
Melt temperature	 <p>Colder Melt</p> <p>A diagram showing a mold cavity with a thermometer indicating a lower temperature, labeled as colder melt.</p>	 <p>Hotter Melt</p> <p>A diagram showing a mold cavity with a thermometer indicating a higher temperature, labeled as hotter melt.</p>
Mold-wall (coolant) temperature	 <p>Colder Coolant Temperature</p> <p>A diagram showing a mold wall with a thermometer indicating a lower temperature, labeled as colder coolant temperature.</p>	 <p>Hotter Coolant Temperature</p> <p>A diagram showing a mold wall with a thermometer indicating a higher temperature, labeled as hotter coolant temperature.</p>

Ram speed	<p data-bbox="581 115 885 157">Improper Ram Speed</p>  <p>The diagram shows a cross-section of a mold cavity. A gray plunger is positioned at the top, with four red arrows pointing left towards the mold walls. The mold cavity is filled with a material that has a jagged, irregular interface between the material and the walls, indicating poor compaction.</p>	<p data-bbox="1010 115 1313 157">Optimized Ram Speed</p>  <p>The diagram shows a cross-section of a mold cavity. A gray plunger is positioned at the top, with three red arrows pointing left towards the mold walls. The mold cavity is filled with a material that has a smooth, uniform interface between the material and the walls, indicating good compaction.</p>
MATERIAL SELECTION		
Melt flow index	<p data-bbox="604 493 863 535">Low Index Material</p>  <p>The diagram shows a melt flow index (MFI) test setup. A gray plunger is positioned at the top of a vertical tube. The tube is filled with a material. A red arrow points downwards from the bottom of the tube, indicating the flow of the material. The flow is relatively narrow and short.</p>	<p data-bbox="1036 472 1302 514">High Index Material</p>  <p>The diagram shows a melt flow index (MFI) test setup. A gray plunger is positioned at the top of a vertical tube. The tube is filled with a material. A red arrow points downwards from the bottom of the tube, indicating the flow of the material. The flow is wider and longer than the one in the low index material diagram.</p>

Filling pattern

What is the filling pattern?

The filling pattern is the transient progression of the melt front within the delivery system and mold cavities. It plays an important role in determining the quality of the part.

Melt front velocity influences filling pattern

► Constant MFV

The ideal filling pattern has melt reaching every extremity of the cavity simultaneously, with a constant melt-front velocity (MFV) (see [Chapter 3](#)) throughout the process. Otherwise, localized over-packing at prematurely filled regions might arise within the part.

► Variable MFV

A variable MFV during filling also leads to changes in the molecular or fiber orientation that immediately freeze at the part surface region when the molten plastic contacts the cold mold.

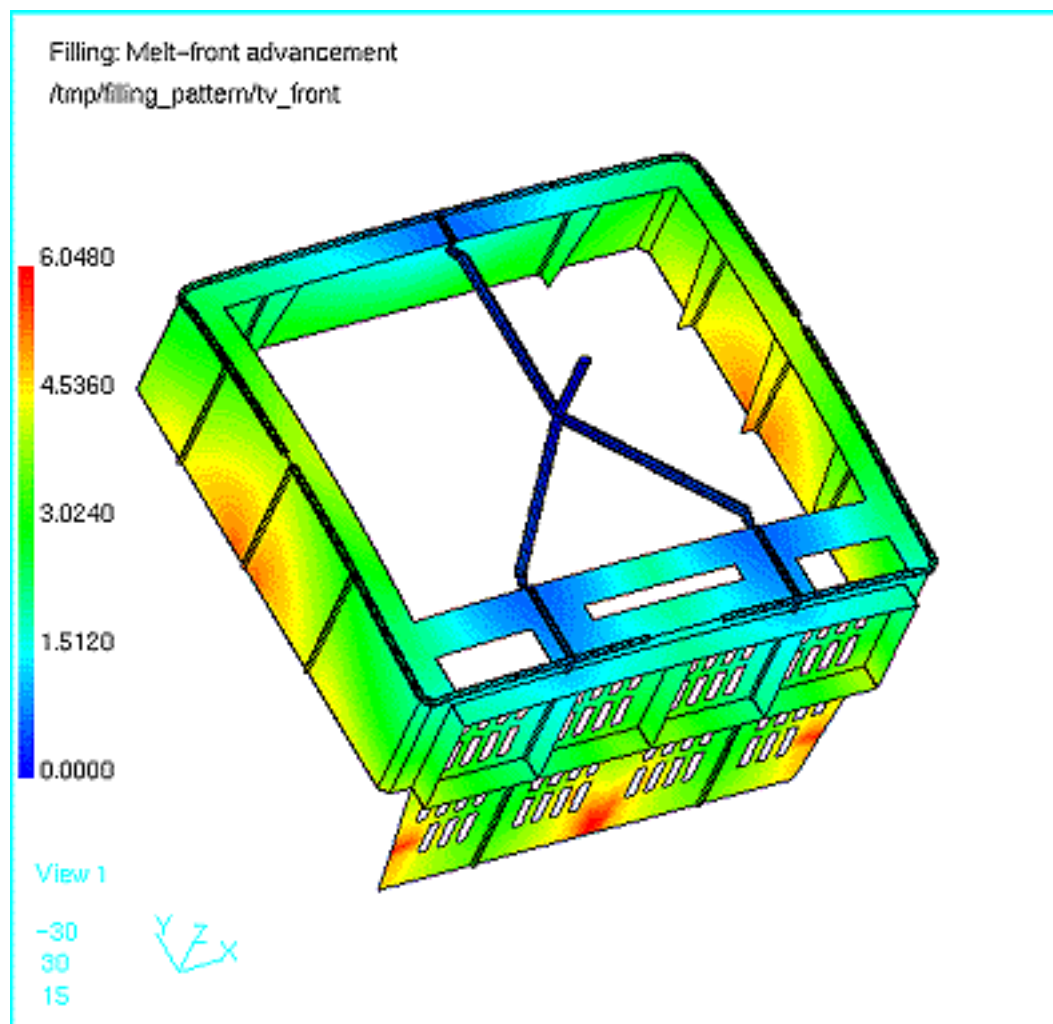


FIGURE 1. Computer-simulated filling pattern in a color-shaded image

MFA and MFV

Definition

Here we present two simple yet important design and process parameters: melt-front velocity (MFV) and melt-front area (MFA). As its name suggests, **melt-front velocity** is the melt-front advancement speed. **Melt-front area** is defined as the cross-sectional area of the advancing melt front: either the length of the melt front multiplied by the thickness of the part (see the diagram below), the cross-sectional area of the runner, or a sum of both, if the melt is flowing in both places. At any time, the product of local MFV and MFA along all moving fronts is equal to the volumetric flow rate.

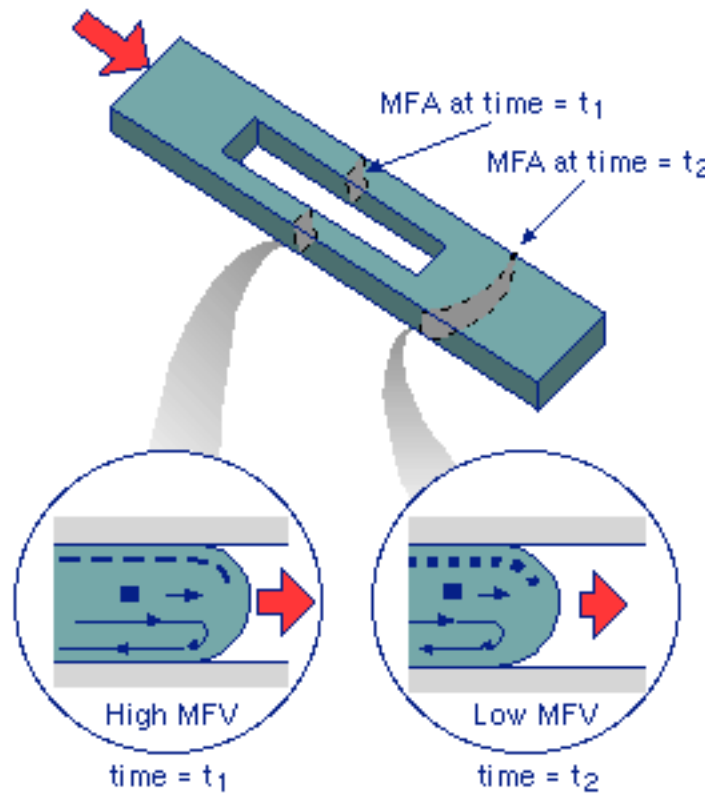


FIGURE 1. Melt-front velocity and melt-front area. Note that a constant volumetric flow rate does not necessarily guarantee a constant velocity at the advancing melt front, due to the variable cavity geometry and filling pattern. With a variable MFV, the material element (shown in square) will stretch differently, resulting in differential molecular and fiber orientations.

Varying MFV

For any mold that has a complex cavity geometry, a constant ram speed (or, equivalently, a constant volumetric flow rate) does not necessarily guarantee a constant velocity at the advancing melt front. Whenever the cross-sectional area of the cavity varies, part of the cavity may fill faster than other areas. The figure above shows an example where the MFV increases around the insert, even though the volumetric flow rate is constant. This creates high stress and orientation along the two sides of the insert and potentially results in differential shrinkage and part warpage.

Equation

The relationship of volumetric flow rate, MFA, and an averaged MFV can be expressed as:

$$\text{Melt-front velocity (MFV)} = \frac{\text{Volumetric injection flow rate}}{\text{Melt-front area (MFA)}}$$

How flow dynamics affect orientation

During the filling stage of the injection molding process, the polymer molecules and reinforcing fibers (for fiber-filled polymers) will orient in a direction influenced by the shear flow. Since the melt is usually injected into a cold mold, most of the orientation in the vicinity of the part surface is almost instantaneously frozen-in, as illustrated below. The state of molecular and fiber orientation depends on the flow dynamics of the melt front and the evolution of fiber orientation. At the melt front, a combination of shear and extensional flows continuously force the fluid elements from the center core to the mold wall, a phenomenon commonly referred to as the "[Fountain flow](#)." Fountain flow behavior greatly influences the molecular/fiber orientations, especially in areas close to the part surfaces.

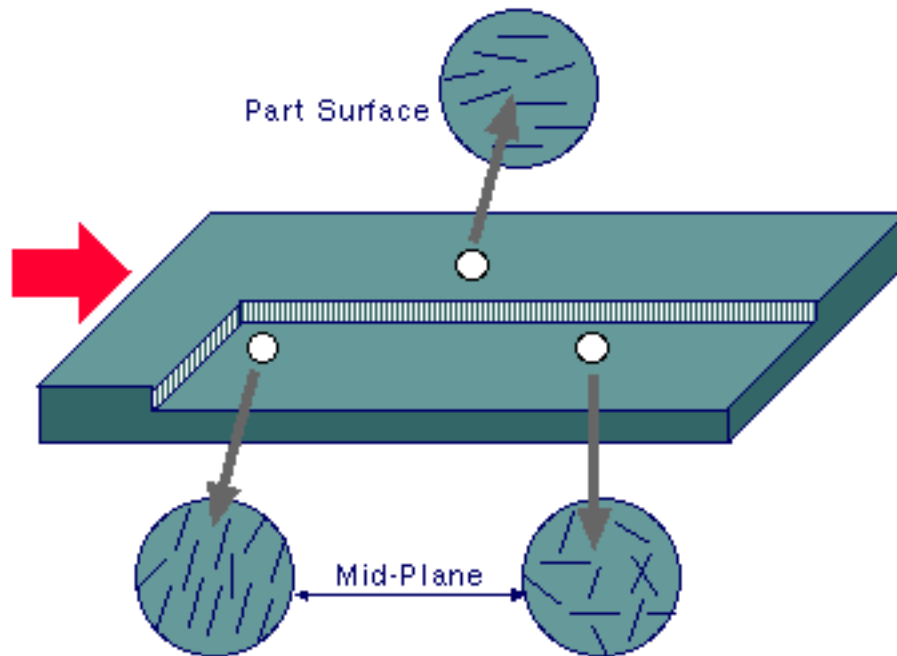


FIGURE 2. Fiber orientations on the part surface and in the center mid-plane of the part

Why constant MFV is important

The dynamics of the melt front are perhaps the least well understood aspect of mold filling, and are beyond the scope of this design guide. However, it is well recognized that the higher the velocity at the melt front, the higher the surface stress and the degree of molecular and fiber orientation. Variable orientation within the part, as a result of variable velocity at the melt front during filling,

leads to differential shrinkage and, thus, part warpage. Therefore, it is desirable to maintain a constant velocity at the melt front to generate uniform molecular and fiber orientation throughout the part.

Flow balance

MFV and MFA are important design parameters, especially for balancing the flow during cavity filling. For example, MFA can be used as an index to quantitatively compare the degree of flow balance. More specifically, when the flow is unbalanced, portions of the melt front reach the end of the cavity while other portions are still moving. The melt-front area changes abruptly whenever such an unbalanced situation occurs. On the other hand, balanced flow generally has a minimum variation of melt-front area in the cavity. For a given complex part geometry, you can determine the optimized gate location by minimizing the variation of MFA in the cavity. As an example, the diagram below shows the MFAs that correspond to a balanced and an unbalanced filling pattern.

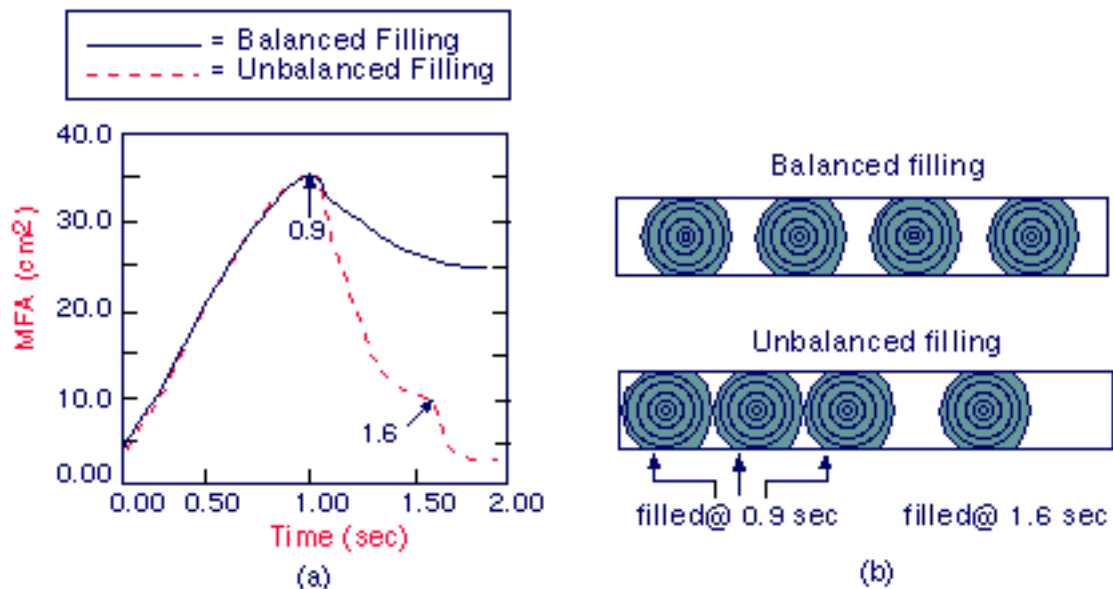


FIGURE 3. a) Variation of MFA with balanced and unbalanced flows, and (b) the corresponding filling patterns.

Material properties for part design

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 1. Typical loading/operating conditions together with the relevant material properties

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

Stress-strain behavior

Part strength

The stress-strain behavior of a material determines the material contribution to part strength (or stiffness), the relationship between load and deflection in a plastic part. Other factors that affect 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.

Tensile properties

It is important to consider the relevant stress-strain behavior that corresponds to the primary (and, commonly, the multiple) load state(s) at the operation temperature and strain rate. However, because of the inherent accuracy problems regarding the current testing procedures for non-tensile 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.

The figure below depicts the tensile bar test sample and the deformation under a pre-set, constant load. The stress (σ) and strain (ϵ) are defined as:

$$\text{stress } (\sigma) \equiv \frac{\text{loading force } (F)}{\text{cross-sectional area } (A)}$$

$$\text{strain } (\epsilon) \equiv \frac{L - L_0}{L_0}$$

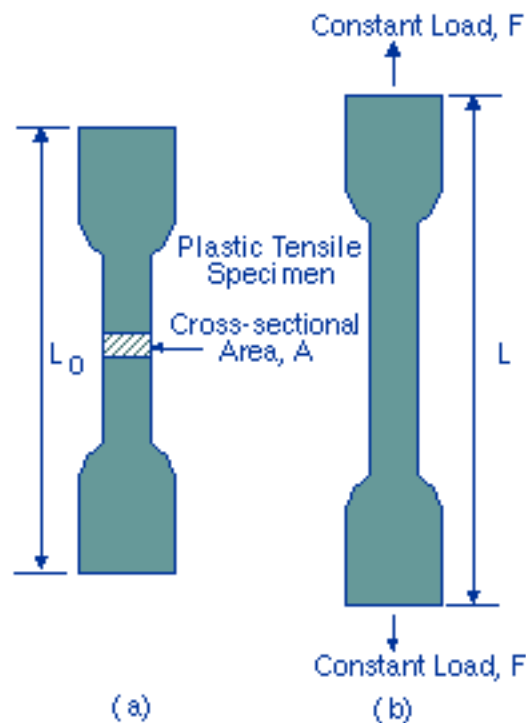


FIGURE 1. (a) The tensile test bar with a cross-sectional area, A , and original length L_0 . (b) Tensile test bar under a constant loading, F , with elongated length, L .

Viscoelastic behavior and spring/dash-pot model

For viscoelastic material such as plastics, the short-term tensile test data tend to reflect values that are predominantly affected by the elastic response of the plastic. 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 dash-pots 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 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 is dependent on the spring constant.

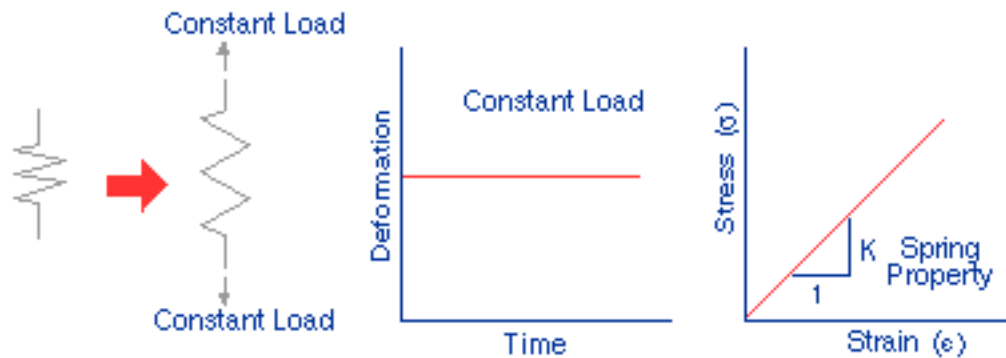


FIGURE 2. A spring represents the elastic response to load.

► Dash-pots represent viscous response to load

The dash-pot in [Figure 3](#) represents the viscous portion of a plastic's response. The dash-pot 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.

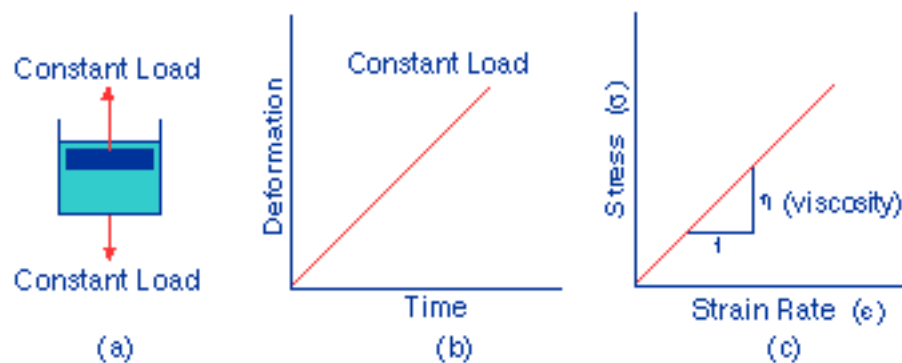


FIGURE 3. A dash-pot represents a viscous response to load.

Voight-Kelvin Mechanical model mimics typical response to load

The Voight-Kelvin Mechanical model, which includes a spring and dash-pot in series with a spring and dash-pot in parallel, is the most common model (see [Figure 4](#)) to mimic the plastics' behavior upon loading.

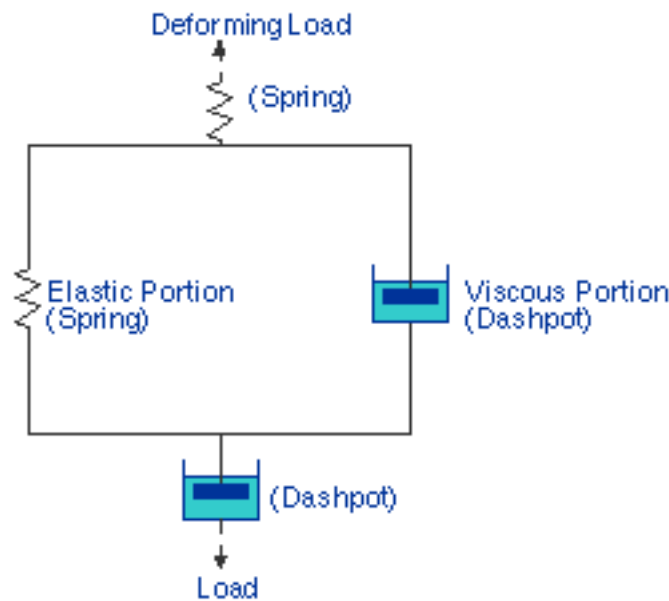


FIGURE 4. A Voight-Kelvin Mechanical model to mimic the typical behavior of plastics' response to load.

The components of the Voight-Kelvin Mechanical model are:

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

Stress-strain curves for Unfilled polymers

below shows a typical stress-strain curve for short-term loading of a typical unfilled thermoplastic material. [Figure 6](#) depicts the same curve as shown in [Figure 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 [Figures 5](#) and [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 (see [Figure 6](#)).

$$\text{Young's modulus } (E) \equiv \frac{\sigma}{\epsilon}$$

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](#), 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](#), 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 5](#)).

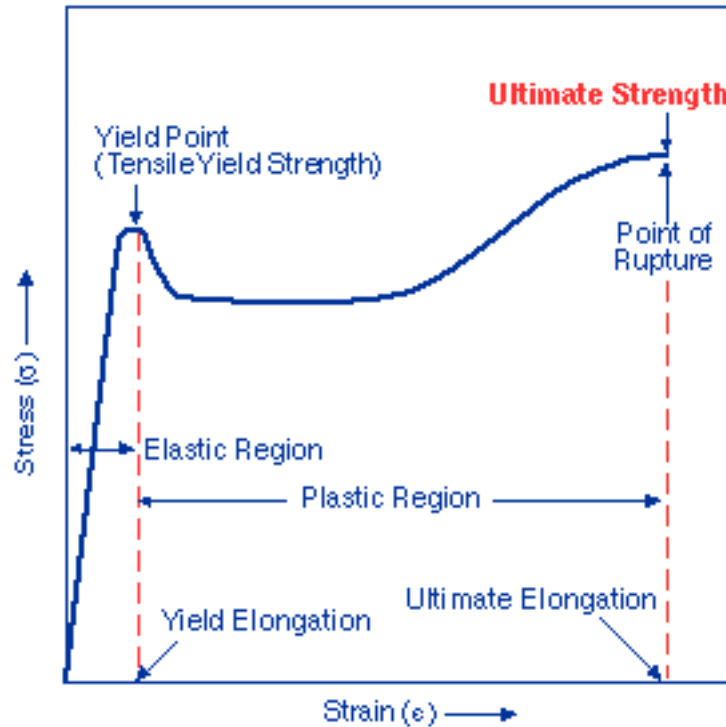


FIGURE 5. Stress-strain curve for a typical thermoplastic.

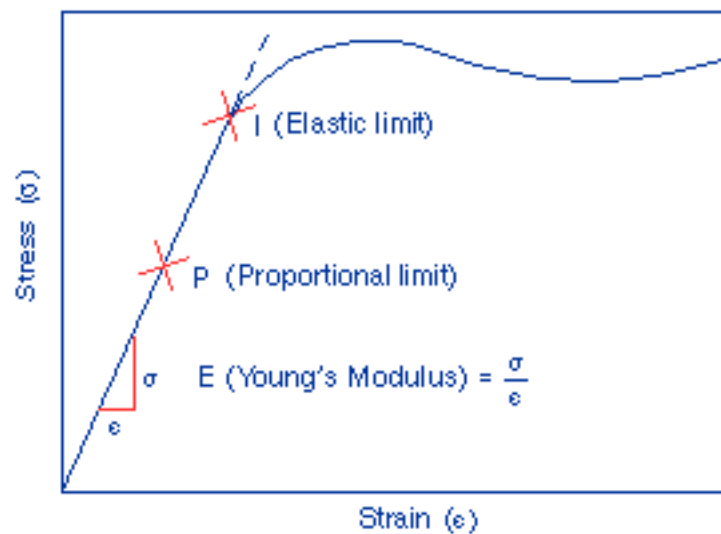


FIGURE 6. The same stress-strain curve as shown in [Figure 5](#), except it is stretched horizontally to show the details within the elastic region. Point P is the proportional limit, most often used as the design strain limit. Point I is the Elastic limit, beyond which the plastic part will not recover its original shape.

Stress-strain curves for fiber-filled polymers

The stress-strain curves for a pair of thermoplastic compounds are shown in [Figure 7](#). The base resin is the same for both compounds, except one compound is unfilled while the other contains 30 percent glass fiber as a 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.

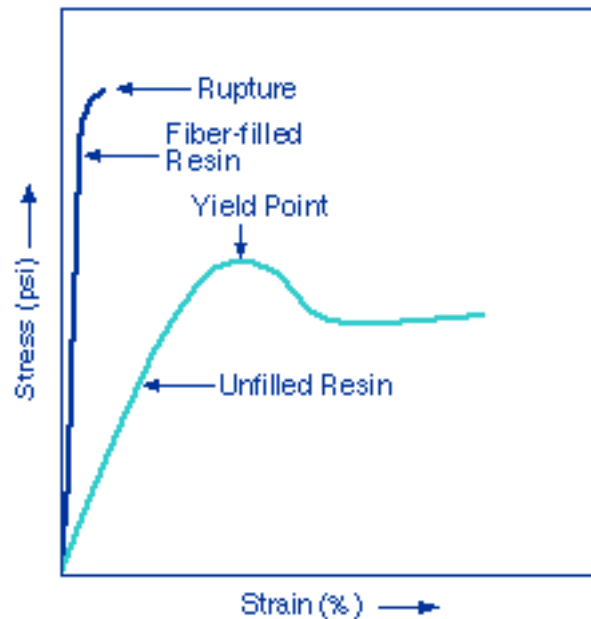


FIGURE 7. Stress-strain curves for a fiber-filled and an unfilled resin.

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 8](#) plots the influence of loading rates and temperature on the tensile stress-strain curve for a semi-crystalline resin. In general, at higher loading rates or lower temperatures, plastics 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 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 semi-crystalline 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.

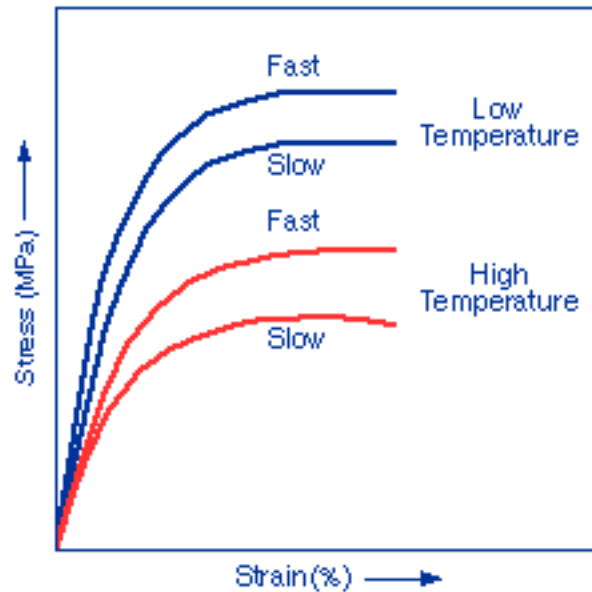


FIGURE 8. Stress-strain curves for a typical polymer at two test temperatures (high and low) and two rates of loading (fast and slow).

Creep and stress relaxation

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

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 9.

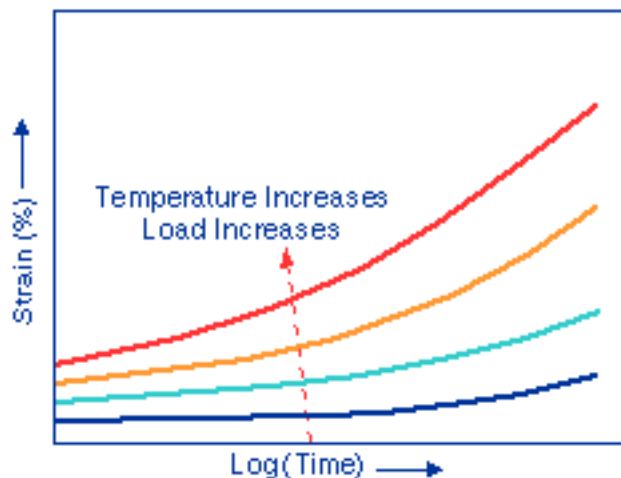


FIGURE 9. A typical creep in flexure curve. Note that the creep is dependent on load and time.

In order to design parts that are subject to long-term loading, designers must use creep data in an

effort 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 10](#). These curves are then used in place of short-term stress-strain curves when designing for applications involving long-term static loading.

Creep modulus

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

$$\text{Creep modulus } (E_c) \equiv \frac{\sigma}{\epsilon(t, T)}$$

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.

Stress relaxation

Stress relaxation is a corollary phenomena 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 10](#) illustrates that at a fixed strain, the stress decreases with the elapsed time.

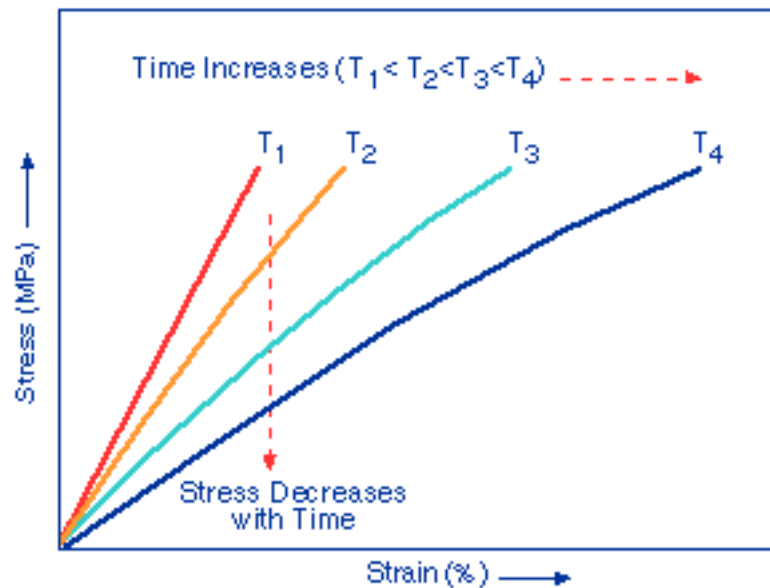


FIGURE 10. These isochronous (fixed-time) curves demonstrate stress relaxation at a constant strain (deflection).

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.

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 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.

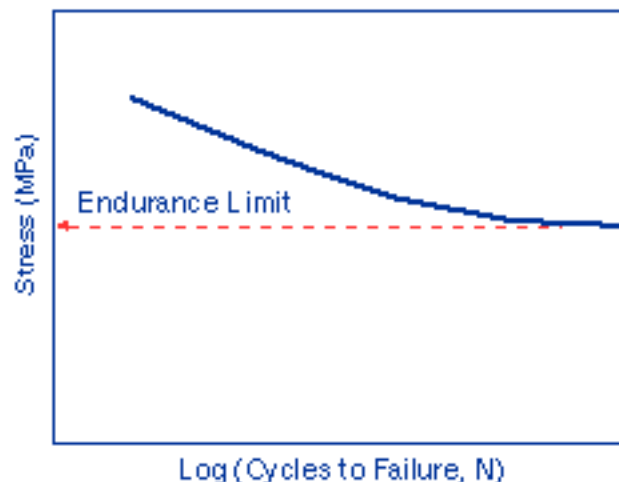


FIGURE 11. A typical flexural fatigue (S-N) curve with the Endurance limit below which the repeated load is unlikely to cause fatigue.

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, micro-cracks on the surface or other physical defects could develop and over time lead to a decrease in overall toughness and eventual failure.

Impact strength

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 (see [High velocity and impact loading](#)). [Figure 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 less tendency to draw and the material acts in a brittle, rather than tough fashion. Decreasing temperature shows a similar behavior, namely, at lower temperatures the plastics are more brittle.

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 due to the effect of stress concentration, as plotted in [Figure 12](#).

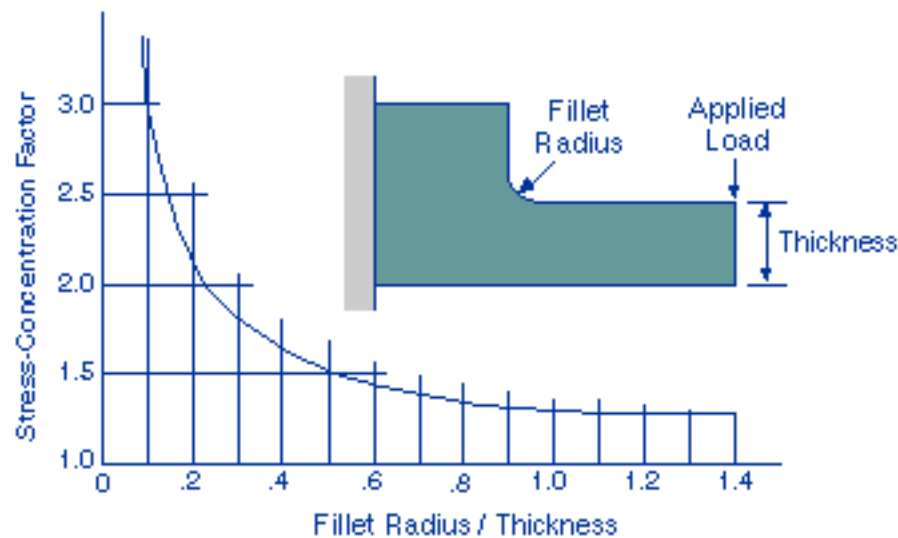


FIGURE 12. Stress concentration as a function of wall thickness and corner radius.

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).

Operation at extreme temperatures

Factors that need to be considered when the operating temperatures are above normal room temperature:

- 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 8](#) shows that strength decreases with increasing temperature.
- Material may exhibit a rubber-like behavior with low modulus and high degree of drawing.

Storage at extreme temperatures

Factors that need to be considered for long-term storage at elevated temperatures:

- 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 that need to be considered when the storage temperatures are below room temperature:

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

Coefficient of thermal expansion

The coefficient of thermal expansion measures the change in dimension from a specific temperature rise. The typical values ($\sim 10^{-4}$ 1/deg-K) are 5 to 10 times larger than that of metals. If the plastics part is rigidly joined to a metal part, the weaker plastic part will fail due to differential expansion or contraction. Depending on the strength of the plastic and the temperature rise, the failure may be immediate or delayed (see [Rate- and temperature-dependency of stress-strain curves](#)). 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.

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

Design for strength

Predicting part strength

The success of failure of the plastics part 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.

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 [Material properties for part design](#).

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](#), are often used to increase the part strength. Fiber reinforcements, oriented in a favorable direction, can also increase the part strength. You should consider using ribs for parts with large spans. Increasing the rib height and/or decreasing the spacing (span) between the ribs also improves part strength.

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 [Fasteners](#).

► 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.

Repeated loading

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

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

Read through the following suggestions if the part you're designing will need to withstand repeated loadings, like the ones given above.

▶ **Cyclic loadings**

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.

▶ **Repeated loadings**

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.

▶ **Avoid micro cracks**

Smooth surfaces, as produced by highly polished mold surfaces, reduce the tendency for micro cracks to form.

▶ **Avoid stress concentration**

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

▶ **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.

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.

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:

- 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:.

- Refrigeration components
- Shipped in the hold of an airplane

Designing for extreme temperatures

You'll need to design parts to accommodate the changes in temperature they'll be exposed to. The following suggestions should help.

▶ 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. [Design for assembly](#) 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.

Part thickness

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.

Cycle time increases with thickness

Injection-molded plastic parts have to be cooled sufficiently before being ejected from the mold to avoid deformation due to 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.

Thick parts tend to warp

Shrinkage is inherent in the injection molding process. Excessive and non-uniform 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.

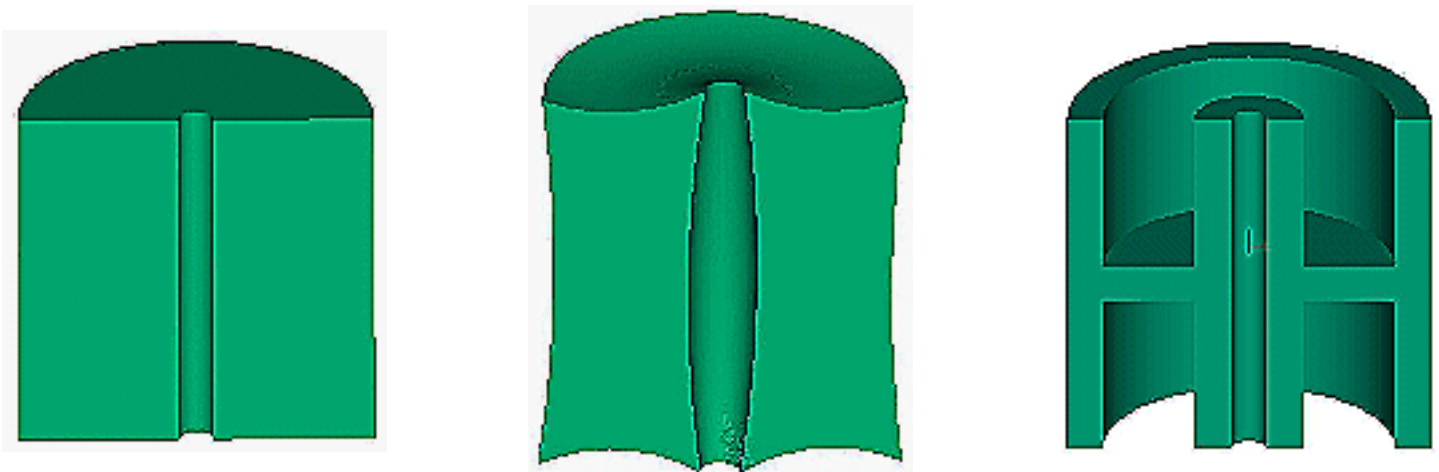


FIGURE 1. A thick part (left) can lead to excessive shrinkage and warpage (center). A recommended alternative design with uniform thickness (right).

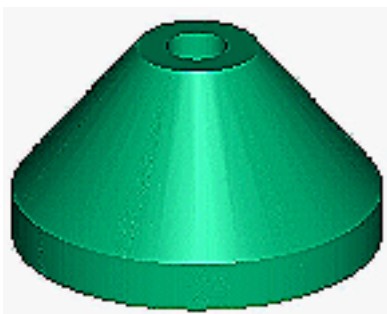
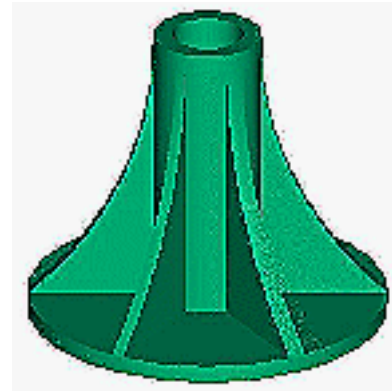
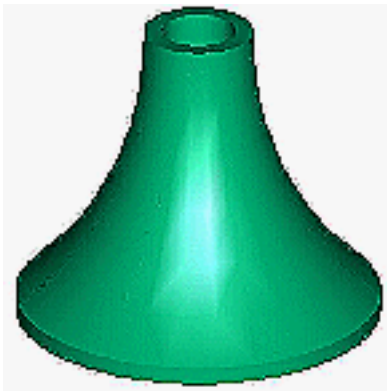
Thin, uniform parts improve surface quality

A combination of thin and heavy part cross-sections can easily produce a "race tracking" effect, which occurs because melt preferentially flows faster along thick sections. Race-tracking leads to [Air traps](#) and weld lines (see [Weld lines and meld lines](#)) that produce defects on part surfaces. In addition, [Sink marks and voids](#) will also arise in thick sections without sufficient packing.

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 of achieving 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. The diagrams shown below provides several examples of designs we recommend vs. designs you should avoid.



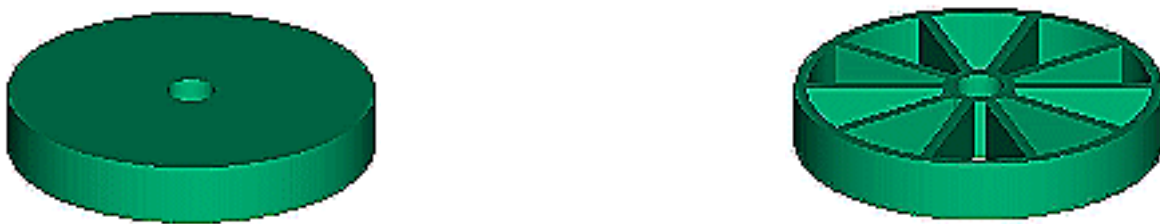


FIGURE 2. Designs with thick sections (left) and alternative designs that use ribs to achieve uniform thickness (right).

Boosting structural integrity with ribs

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.

Typical uses for ribs

- 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.

Designing ribs

Keep [part thickness](#) as thin and uniform as possible. This will shorten the cycle time, improve dimensional stability, and eliminate surface defects. The use of ribs is an effective way of achieving 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.

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 1](#) for recommended design parameters.

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 1](#), 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.

Alternative configurations

As shown in [Figure 2](#), 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.

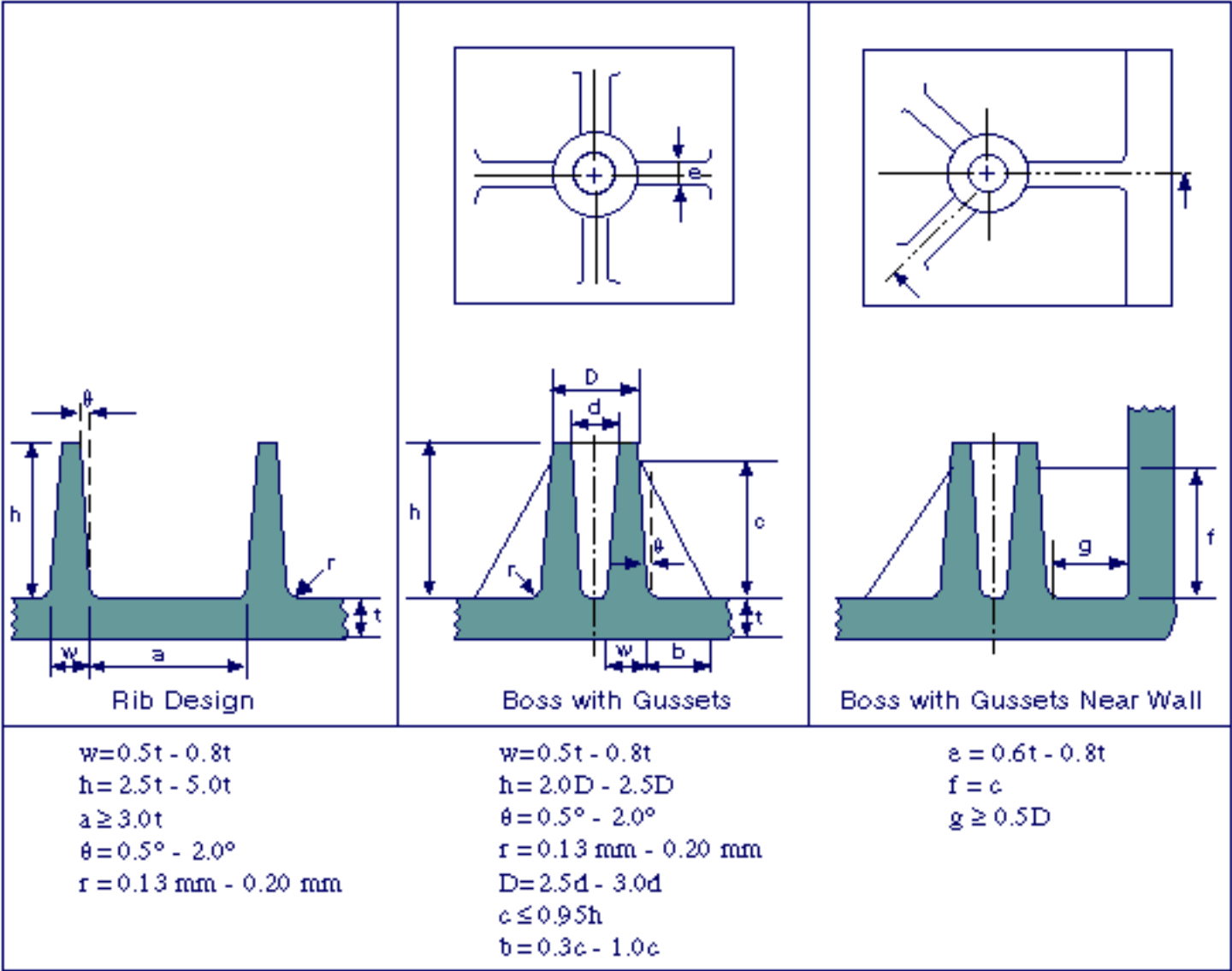


FIGURE 1. Recommendations for rib cross sections.

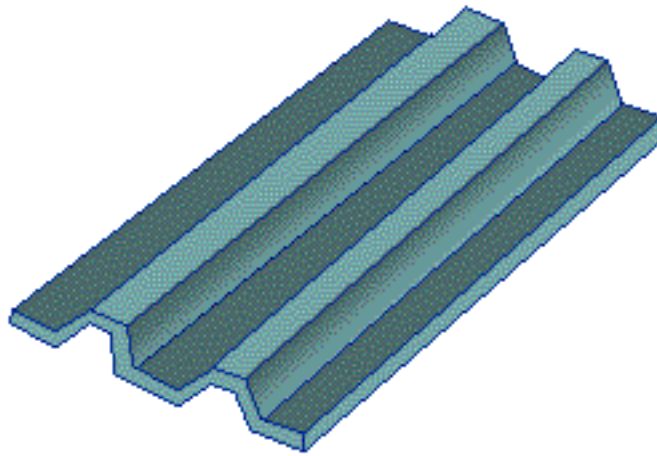


FIGURE 2. Corrugations instead of ribs

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 3. Honeycomb ribbing attached to a flat surface provides excellent resistance to bending in all directions.

Design for assembly

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. However, due to the limitations of the mold and the process, functional requirements, and/or economic considerations, it is still sometimes necessary to mold various components separately and then assemble them together.

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.

Fit between plastics parts

- If the two plastics 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 1](#)).

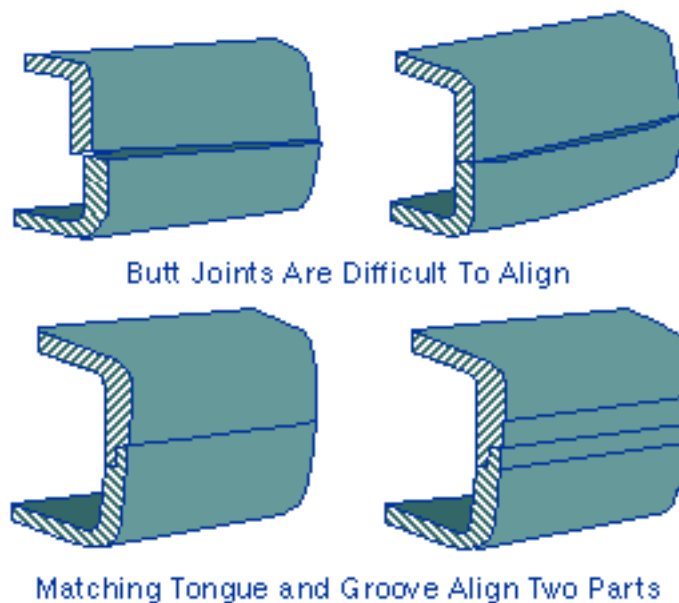


FIGURE 1. Matching half-tongue and groove align the two parts edges, within normal tolerances.

Fit between plastics parts and metal parts

Make sure that the joint between the plastic and metal allows the plastic part to expand without regard to the expansion of the metal part.

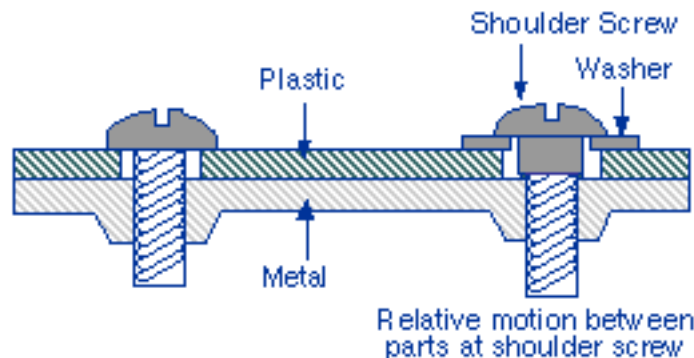


FIGURE 2. Design the joint between plastic and metal to allow for greater thermal expansion and contraction of the plastic. This includes use of shouldered fasteners and clearance between the fastener and the plastic.

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 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.

Interference chart

[Figure 3](#) 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 recommended minimum length of interference is twice the insert diameter.

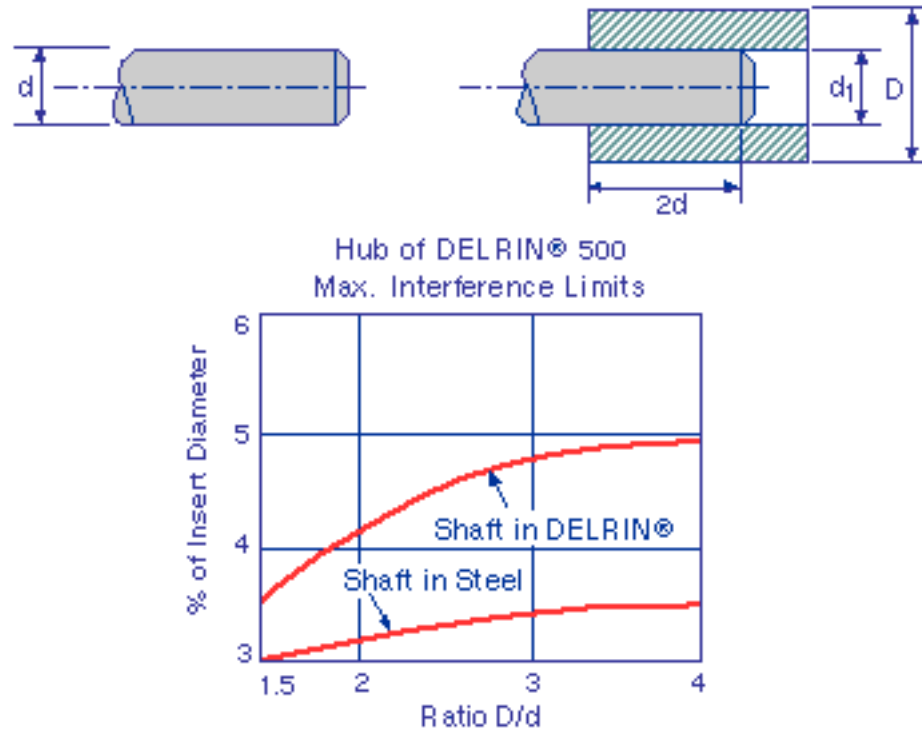


FIGURE 3. Maximum interference limits, pressing a metal shaft into a plastics hub. These curves are specific to the material. The max. interference limit ($d - d_1$) as a percentage of the insert diameter, d , depends on the shaft material and the diameter ratio of the hub and insert (D/d). The recommended minimum length of interference is twice the insert diameter, $2d$.

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 3](#)) can be calculated with the following formula.

$$I = \left(\frac{S_s \times d'}{W} \right) \times \left[\left(\frac{W + \nu_k}{E_k} \right) + \left(\frac{1 - \nu_s}{E_s} \right) \right]$$

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

where:

I = diametrel interference ($d - d_1$), 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

ν_h = Poisson's ratio of hub material

ν_s = Poisson's ratio of shaft material

W = geometry factor

Tolerance

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

Mating metal and plastic parts

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

Snap-fit joints

Snap-fit joints rely on the ability of a plastics 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](#)

Annular snap-fit joints

This is a convenient form of joint for axis-symmetrical 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.

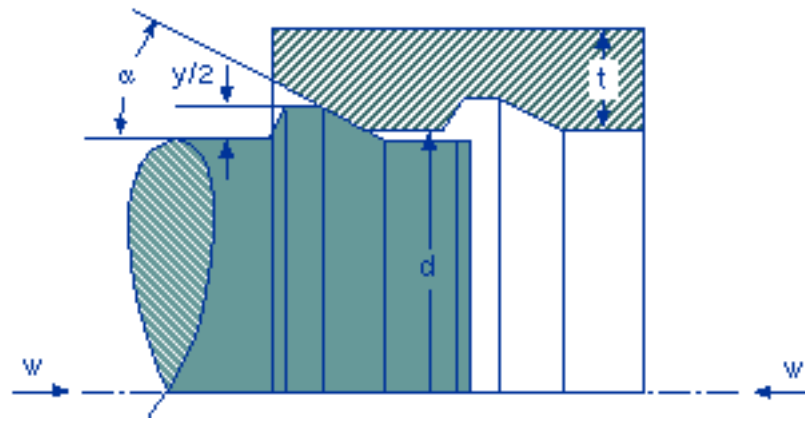


FIGURE 4. Typical annular snap-fit joint. The assembly force, w , strongly depends on the lead angle, α , and the undercut, y , half of which is on each side of the shaft. The diameter and thickness of the hub are d and t , respectively.

Hoop stress

[Figure 5](#) 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, σ , to exceed the proportional limit of the material.

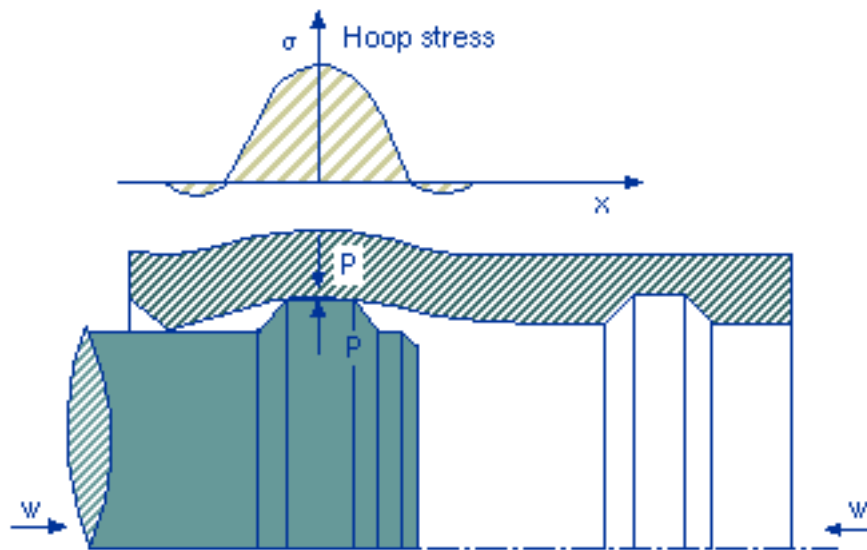


FIGURE 5. Stress distribution during the joining process.

Permissible deformation (undercut)

The permissible deformation (or permissible undercut, y , shown in [Figure 4](#)) 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.

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

► 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 the equation above) 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.

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.

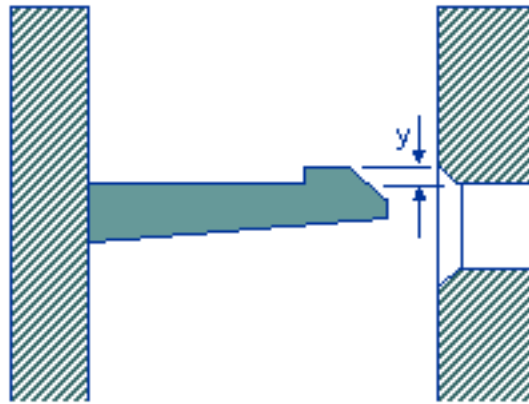


FIGURE 6. Typical cantilever snap-fit joint. The interference between the hole and the hook, y , represents the deflection of the beam as the hook is inserted into the hole.

Proportional limit

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

Designing the hook

Either the width or thickness can be tapered (see [Figure 6](#)). 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

Designing the base

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

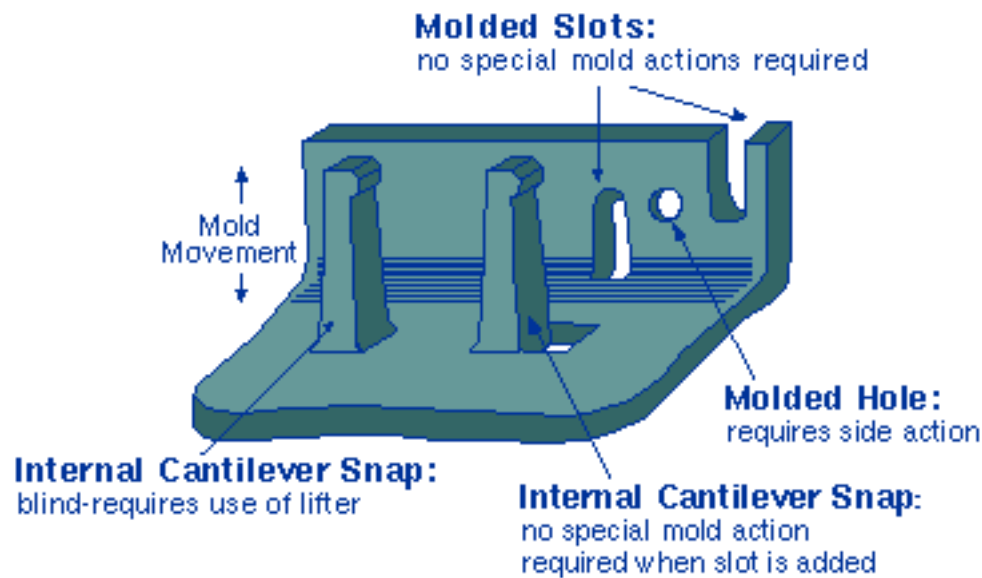


FIGURE 7. Design the snap-fit features for ejection.

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 8](#)) is subject to shear loads. This type of fastener is good for frequent assembly and disassembly.

Design formula

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

$$\sin \phi = \frac{y_1}{l_1} = \frac{y_2}{l_2}$$

where:

Φ = angle of twist

y_1 and y_2 = deflections

l_1 and l_2 = lengths of lever arms (see [Figure 8](#))

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

$$\Phi_{pm} = \frac{180}{\pi} \times \gamma_{pm} \times \frac{l}{r}$$

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:

$$\gamma_{pm} = (1 + \nu) \epsilon_{pm}$$

$$\gamma_{pm} = 1.35 \epsilon_{pm}$$

where:

γ_{pm} = permissible shear strain

ϵ_{pm} = permissible strain

ν = Poisson's ratio (approx. 0.35 for plastics)

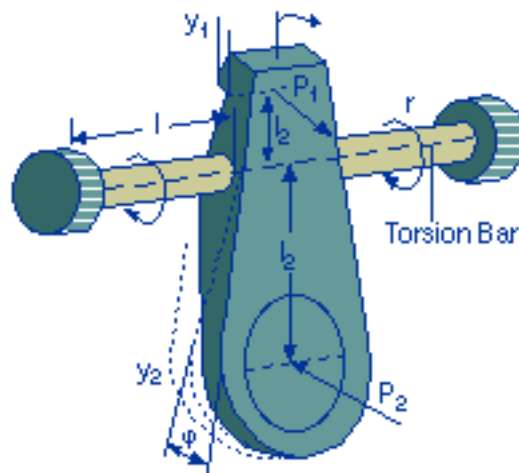


FIGURE 8. Torsional snap-fitting arm with torsional bar. Symbols defined in text above.

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 screw head or nut or on the head of the rivet could act as stress risers and cause early failure.

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 9](#) provides recommendations for the diameter of clearance holes for various screw sizes.

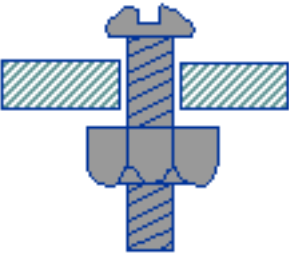
	Screw Size	Hole Diameter (in.)	Hole Diameter (mm)
	#2	.096	2.44
	#4	.122	3.10
	#6	.148	3.76
	#8	.174	4.42
	#10	.200	5.08
	#12	.226	5.74
	1/4	.260	6.60
	5/16	.323	8.20
	3/8	.385	9.78

FIGURE 9. Recommendations for clearance between the machine screw and hole in the plastic. The pan-head style of the screw is recommended.

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, and 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.

Do not use	Since
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 over-tightened and over-stressed. Stress-cracking will result

Molded threads

Molding threads into the plastic component avoids having to use separate fasteners such as screws and rivets. If the threads are molded, 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 10](#) shows how to design an unthreaded lead-in.

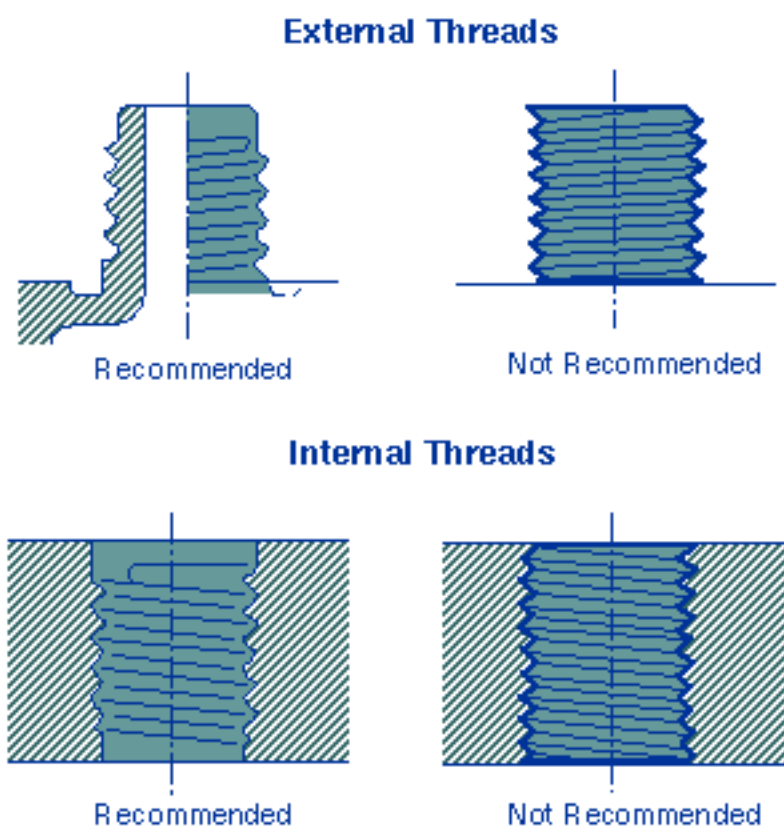


FIGURE 10. Recommended design for molded threads.

Below are some guidelines to designing molded threads:

► 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 un-screwing the mold component from the part, either manually or by action of the mold. Large internal threads can be formed on collapsing mold components.

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.

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.

Support posts

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

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.

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.

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. You'll need to design an energy director, a small triangular raised bead, on one of the faces to be welded.

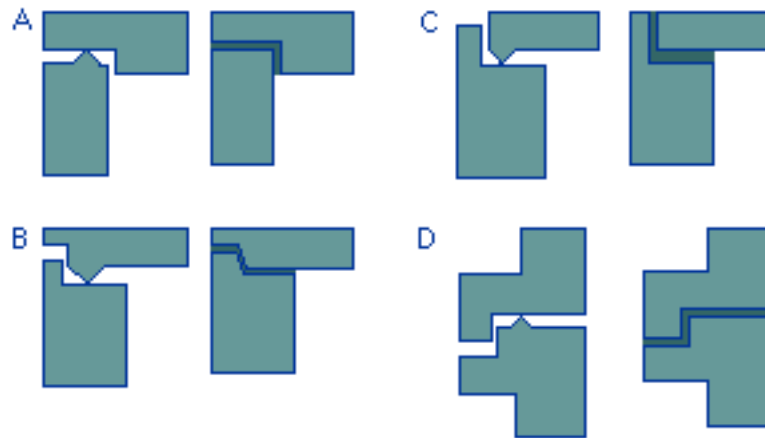


FIGURE 11. Recommendations for the design of ultrasonic welded joints.

Runner systems

Helping the melt flow

A runner system directs the melt flow from the sprue to the mold cavities. Additional pressure is required to push the melt through the runner system. Shear (frictional) heat generated within the melt while the material is flowing through the runner raises the melt temperature, also facilitating the flow.

Runner size considerations

Although properly sizing a runner to a given part and mold design has a tremendous pay-off, it is often overlooked since the basic principles are not widely understood.

► Pros and cons of large runners

While large runners facilitate the flow of material at relatively low pressure requirements, they require a longer cooling time, more material consumption and scrap, and more clamping force.

► Pros and cons of small runners

Designing the smallest adequate runner system will maximize efficiency in both raw material use and energy consumption in molding. At the same time, however, runner size reduction is constrained by the molding machine's injection pressure capability.

► Optimal runner size

C-MOLD Runner Balancing is an excellent software tool for computing the optimal runner size that conveys a balanced filling pattern with a reasonable pressure drop.

Payoffs of good runner design

A runner system that has been designed correctly will:

- Achieve the optimal number of cavities
- Deliver melt to the cavities
- Balance filling of multiple cavities
- Balance filling of multi-gate cavities
- Minimize scrap
- Eject easily
- Maximize efficiency in energy consumption
- Control the filling/packing/cycle time.

Determining the number of cavities

Factors involved

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

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, multi-cavity molds are preferred. The number of cavities is dependent 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:

$$\text{Number of cavities} = L \times K \times tc / tm$$

► Shot capacity

The injection machine shot capacity is also a factor in determining the number of cavities. Take 80 percent of the machine capacity as the shot weight (S) and divide by the part weight (W) to get the number of cavities. The relation is:

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

► Plasticizing capacity

The injection machine plasticizing capacity is also a factor. Divide the plasticizing capacity (P) of the machine by the estimated number of shots per minute (X) and part weight (W). The relation is:

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

Planning the runner system layout

Basic layouts

There are three basic runner system layouts typically used for a multi-cavity system. These layouts are illustrated in [Figure 1](#) below.

- Standard (herringbone) runner system
- "H" bridge (branching) runner system
- Radial (star) runner system

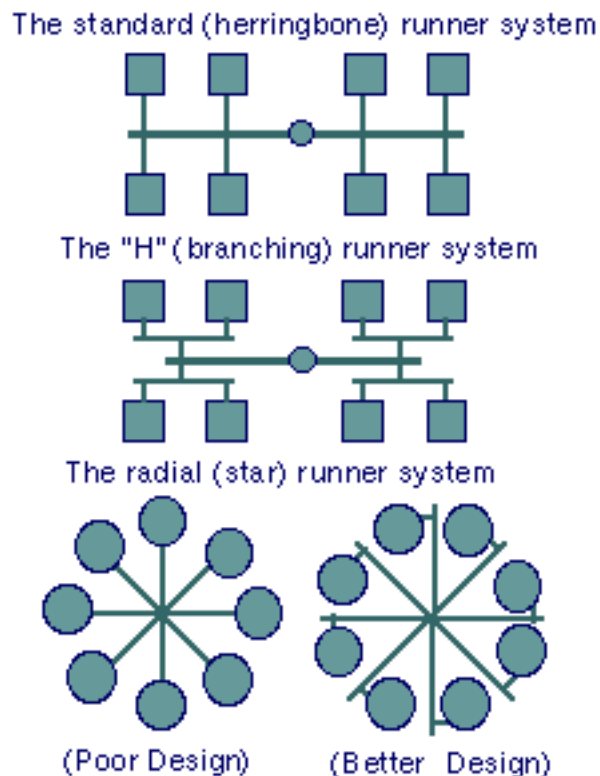


FIGURE 1. Basic runner system layouts

Balanced vs. unbalanced layouts

► Balanced layouts

The "H" (branching) and radial (star) systems are considered to be naturally balanced. The naturally balanced runner provides equal distance and runner size from the sprue to all the cavities, so that each cavity fills under the same conditions.

► Unbalanced layouts

Although the herringbone is naturally 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 and the length of the runner.

► Automatic balancing

Runner balancing can be accomplished automatically with C-MOLD runner balancing analysis.

Determining sprue dimensions

General guidelines

The dimensions of the sprue depend primarily on the dimensions of the molded part and especially its wall thickness. Here are some general guidelines you should consider:

- The sprue must not freeze before any other cross section. This is necessary to permit sufficient transmission of holding pressure.
- The sprue must de-mold easily and reliably.

Example

[Figure 1](#) below illustrates a sprue designed under these guidelines. Use the following formulas to design the sprue. A radius r_2 is suggested at the root of the sprue to avoid the sharp corner between the sprue and molding and to facilitate the flow of material.

$$D_{co} \geq t_{max} + 1.5 \text{ mm}$$

$$D_s \geq D_n + 1.0 \text{ mm}$$

$$\alpha \geq 1^\circ - 2^\circ$$

$$\tan \alpha = D_{co} - D_s / 2L$$

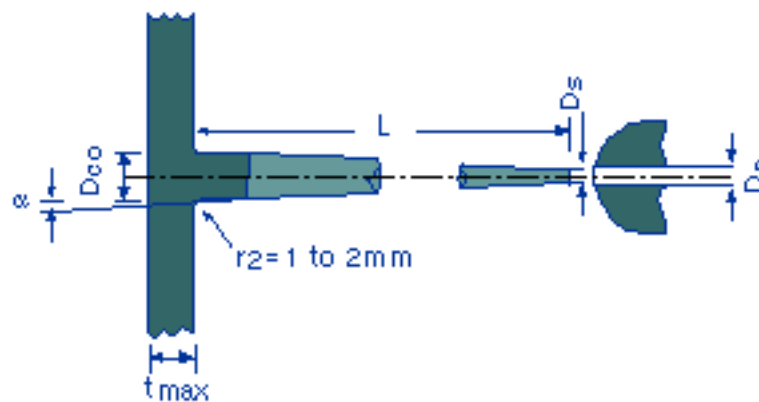


FIGURE 1. Radius at sprue base improves processing

Designing runner cross sections

Common designs

There are several common runner cross-sectional designs. They are illustrated in [Figure 1](#).

- Full-round runner
- Trapezoidal runner
- Modified trapezoidal runner (a combination of round and trapezoidal runner)
- Half-round runner
- Rectangular runner

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 semi-circular 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.

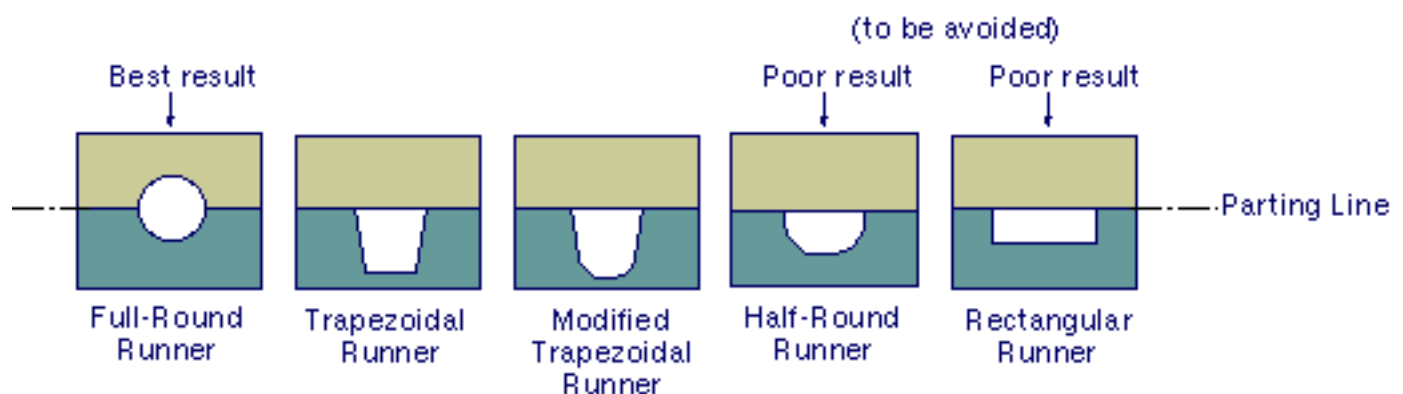


FIGURE 1. Commonly used runner cross sections

Hydraulic diameter and flow resistance

To compare runners of different shapes, you can 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:

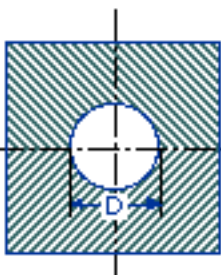
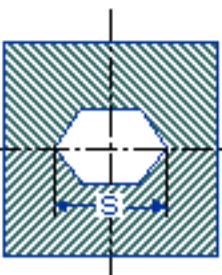
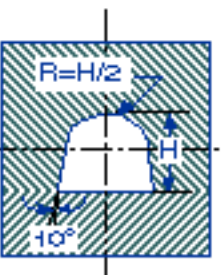
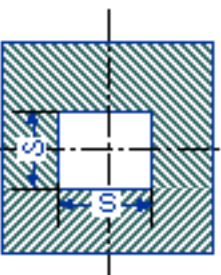
$$D_k = \frac{4A}{P}$$

where D_k = hydraulic diameter

A = cross section area

P = perimeter

Figure 2 illustrates how to use the hydraulic diameter to compare different runner shapes.

Cross Section				
D_h	D	$0.9523D$	$0.9116D$	$0.8862D$

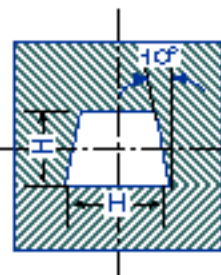
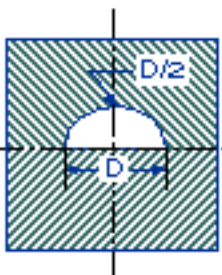
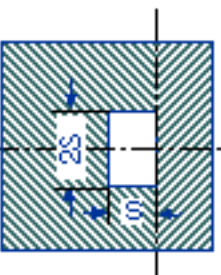
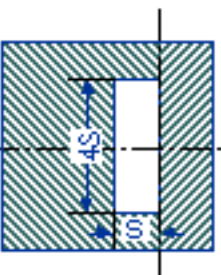
Cross Section				
D_h	$0.8771D$	$0.8642D$	$0.8356D$	$0.7090D$

FIGURE 2. Equivalent hydraulic diameters

Determining runner dimensions

Factors you'll need to consider

The diameter and length of runners influence flow resistance. The higher the flow resistance in the runner, the higher the pressure drop will be. Reducing flow resistance in runners by increasing the diameter will use more resin material and cause a longer cycle time if the runner has to cool down before ejection. First design the diameter by using empirical data or the following equation. Then fine-tune the runner diameter using C-MOLD to optimize the delivery system.

Formula

Following is the formula for runner dimension design:

where

D = runner diameter (mm)

W = part weight (g)

L = runner length (mm)

$$D = \frac{w^{1/2} \times L^{1/4}}{3.7}$$

where D = runner diameter (mm)

W = part weight (g)

L = runner length (mm)

Example: using empirical data to calculate runner dimensions

Figures 1 and 2 provide empirical data that you can use to calculate runner dimensions. For example, what should the runner diameter be for an ABS part of 300 grams, with a nominal thickness of 3 mm, and a runner length of 200 mm?

- 1 According to Figure 1, take the point of 300 grams of the ordinate, draw a horizontal line and meet the line of nominal thickness = 3 mm, draw a vertical line through the intersection point and meet the abscissa at 5.8 mm.

This is the reference diameter.

- 2 Using Figure 2, take the point of 200 mm of the coordinate, draw a horizontal line and meet the curve, draw a vertical line through the intersection point and meet the abscissa at 1.29.

This is the length coefficient.

3Multiply 5.8 mm by 1.29 to calculate the runner diameter.

The diameter is 7.5 mm.

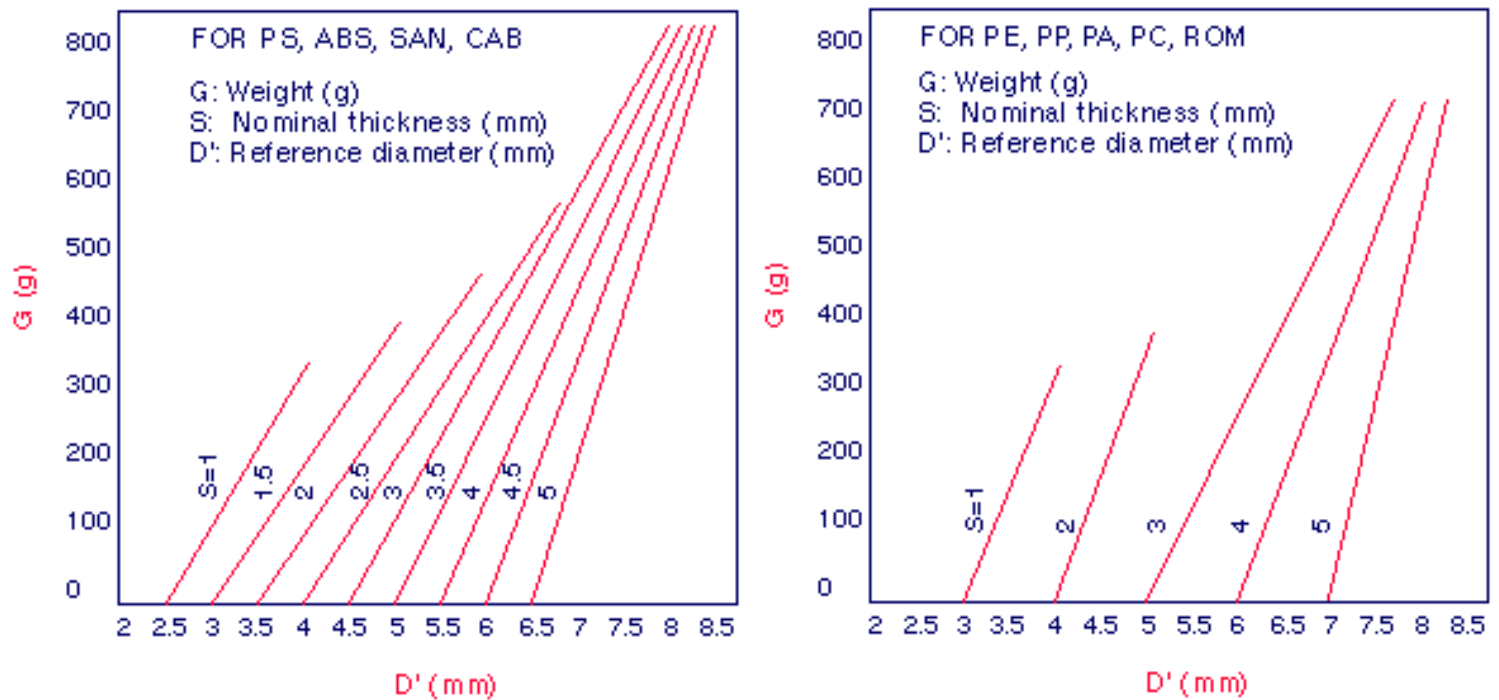


FIGURE 1. Runner diameter chart for several materials. G=weight (g); S=nominal thickness (mm); D=reference diameter (mm).

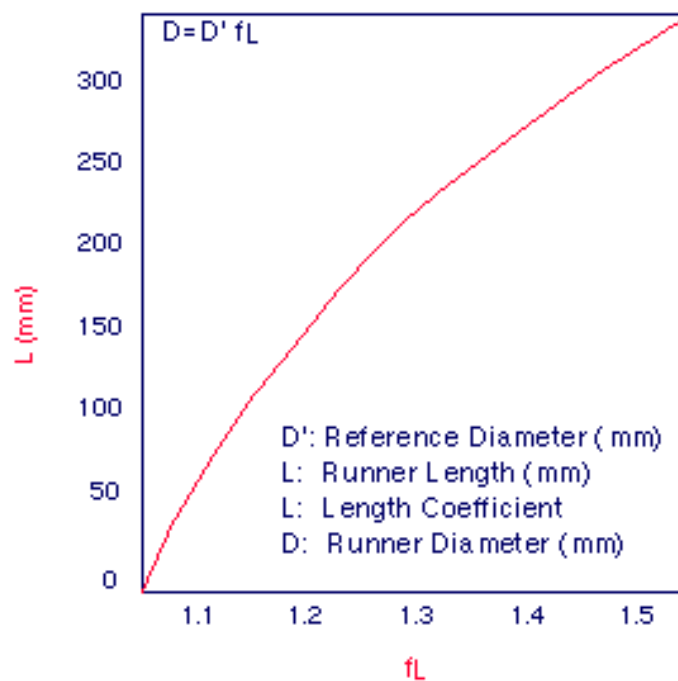


FIGURE 2. Effect of runner length and length coefficient on diameter.

Typical runner diameters

Typical runner diameters for unfilled materials are listed in [Table 1](#).

TABLE 1. Typical runner diameters for unfilled generic materials

Material	Diameter		Material	Diameter	
	mm	inch		mm	inch
ABS, SAN	5.0-10.0	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	5.0-10.0	3/16-3/8	Polyamide	5.0-10.0	3/16-3/8
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
Ionomers	2.0-10.0	3/32-3/8	Polystyrene	3.0-10.0	1/8-3/8
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	3/16-3/8	Polyurethane	6.0-8.0	1/4-5/16

Hot runner systems

Overview

The ideal injection molding system delivers molded parts of uniform density, and free from all runners, flash, and gate stubs. To achieve this, a hot runner system, in contrast to a cold runner system, is employed. The material in the hot runners is maintained in a molten state and is not ejected with the molded part. Hot runner systems are also referred to as hot-manifold systems, or runnerless molding.

Types of hot runner systems

There are two types of hot runner systems:

► 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.

► 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. The second consists of a cartridge-heated manifold with 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 1](#) lists advantages and disadvantages of the three hot runner systems, which are sketched in [Figure 1](#).

TABLE 1. Advantages and disadvantages of hot-runner systems

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

Externally Heated

- Improved distribution of heat.
- Better temperature control.

- Higher cost and complicated design.
- Should take into account thermal expansion of various mold components.

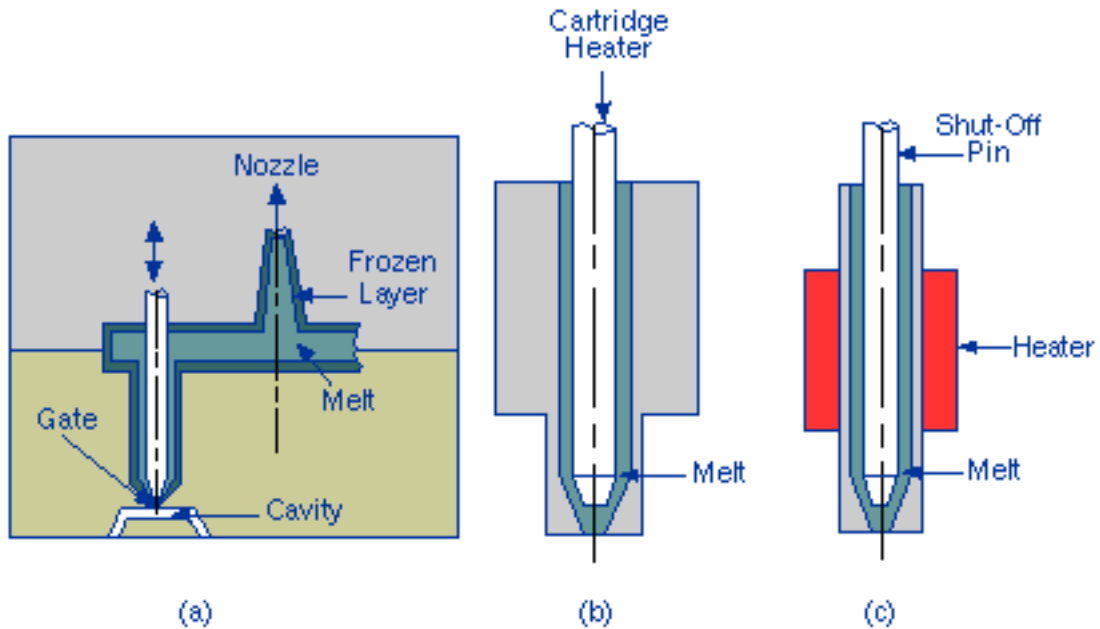


FIGURE 1. Hot runner system types: (a) the insulated hot runner, (b) the internally heated hot-runner system, and (c) the externally heated hot-runner system.

Runner balancing

Importance of balanced flow

Balanced flow into the cavities is a prerequisite for a quality part. This can be achieved by changing the runner size and length. Changing the gate dimension may give a seemingly balanced filling. However, it affects the gate freeze-off time greatly, which is detrimental to part uniformity. Whenever possible, a naturally balanced runner system should be used to balance the flow of material into the cavities. If a naturally balanced runner is not possible, then the runner system should be artificially balanced, as shown below. Using C-MOLD's Runner Balancing tool to artificially balance a runner system not only saves time and expense, but also greatly improves the uniform filling of the part.

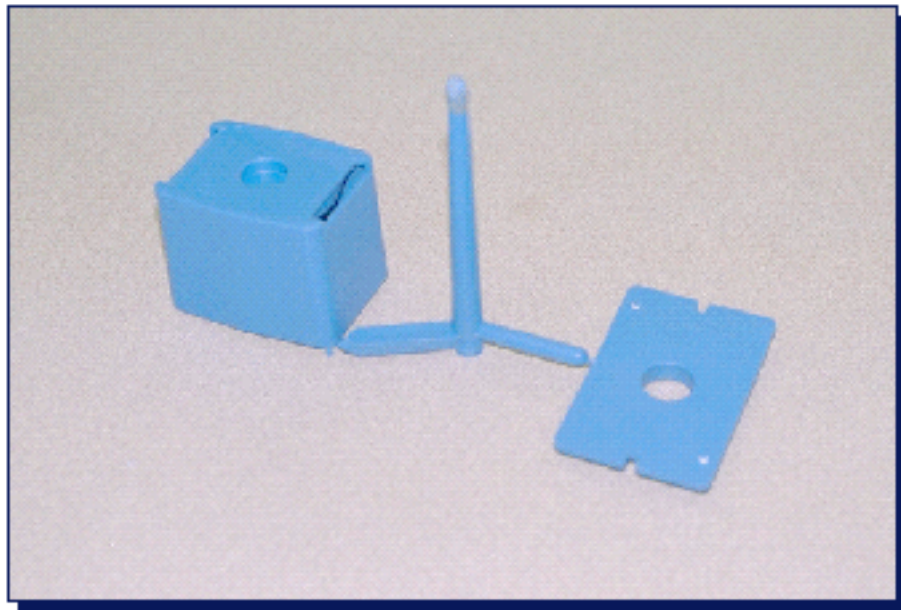


FIGURE 1. A family-mold part with artificially balanced runners

Reducing runner diameter

To balance a runner system, encourage flow to the cavities farthest from the sprue by reducing the diameter of runners feeding the other cavities. Note that decreasing the runner diameter too much may cause it to freeze prematurely, causing a short shot. On the other hand, increased frictional shear heating may actually reduce the resin's viscosity, and thus, resistance to flow and fill the cavity even faster. Keep in mind that non-standard runner diameters will increase mold manufacturing and maintenance costs.

Using tighter process controls

An artificially balanced runner system designed for one material may not work for others. Further, an artificially balanced runner system requires tighter process controls. A small variation in the process control will alter the filling pattern of the mold, leading to consistently unbalanced filling.

In the course of finalizing a runner design, C-MOLD analysis can help identify the sensitivity of the design to flow rate and the appropriate process window.

Varying the injection speed

For example, if you use a standard (herringbone) runner system with various injection rates, then various filling patterns will result. Generally speaking, a slow injection rate will first fill parts farther out onto the runner, while a faster injection rate will first fill the parts closest to the sprue. This is because at a slower injection rate, the melt tends to hesitate at the restricted gate it first encounters. It moves out to fill the remaining runner system. By the time all the runner branches are filled, the melt at the first, upstream gates may have already become more resistant than the downstream gates, due to solidification. Varied injection speed will result in filling patterns between these two extremes, as illustrated in [Figure 2](#).

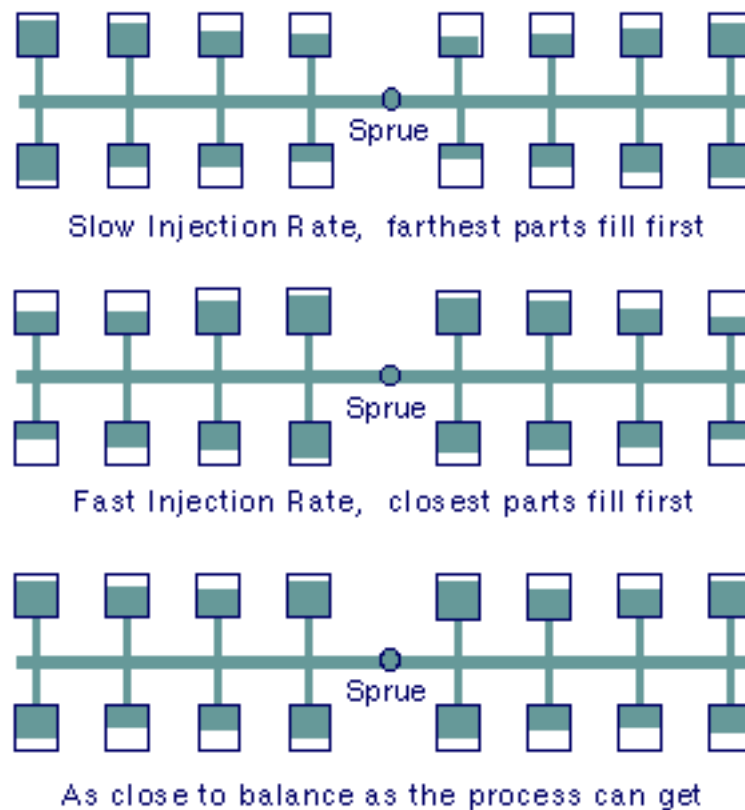


FIGURE 2. Filling patterns resulting from various injection rates, in an unbalanced runner system.

Design rules

Overview

Runner design dictates part quality and productivity. The design rules that follow provide general guidelines for runner design.

Runner size

▶ Runner cross section

The cross-sectional area of a runner should not be smaller than that of the sprue, to permit rapid, unaltered flow of the material to the gating area.

▶ Cold runner diameter

The selection of a cold runner diameter should be based on standard machine tool cutter sizes. The minimum recommended runner diameter for most materials is 1.5 mm (0.06 inches). Typical runner diameters for various unfilled generic materials are shown in [Typical runner diameters for unfilled generic materials](#).

See [Determining runner dimensions](#) for more information on this topic.

▶ Trapezoidal runners

The depth of a trapezoidal runner is approximately equal to its width, with a 5° to 15° draft (taper) on each side wall.

▶ Scrap

Use a small runner size to minimize scrap volume.

Raising melt temperature

Smaller runner size is preferred over larger runner size in order to generate viscous (frictional) heating. Viscous heating generated in the runner system is an effective way to raise the melt temperature instead of using a high barrel temperature, which is likely to cause material degradation.

Branched runners

Each time a runner is branched, the diameter of the branch runners should be smaller than the main runner, because less material flows through the branches and it is economically desirable to use minimum material in the runners.

Where N is the number of branches, the relationship between the main runner diameter (d_{main}) and the branch runner diameter (d_{branch}) is

$$d_{main} = d_{branch} \times N^{1/3}$$

Runner intersections

All runner intersections should have a cold solid slug well to help the flow of material through the runner system and into the cavity. the diagram below shows that the length of the well is usually equal to the runner diameter.

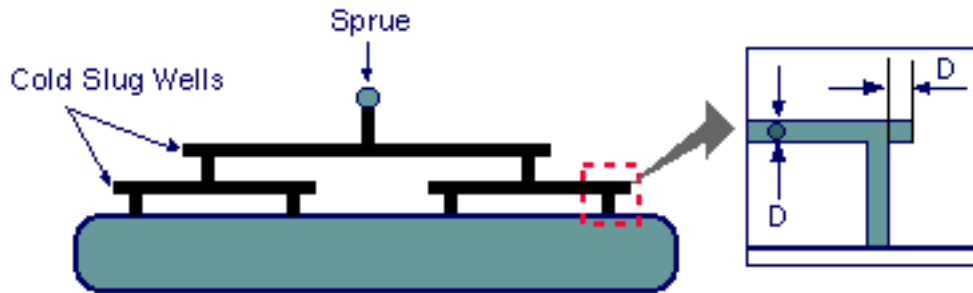


FIGURE 1. The cold slug well is typically an extension of the runner at an intersection with another runner.

Hot runners

When designing Hot runner systems, it is necessary to consult suppliers for availability of the proper manifold and drops.

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.

Gate design overview

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.

Single vs. multiple gates

You'll usually have better success with a single gate, unless the length of the melt flow exceeds practical limits. Multiple gates always create weld and meld lines where the flows from the separate gates meet. Except for long, narrow parts, a single gate into the body of the part (as opposed to an edge gate) will assure more uniform distribution of material, temperatures, and packing, and better orientation effects. While a single gate into the body of the part might incur a higher initial tool cost, lower scrap rates and higher part quality will quickly justify this expense.

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 usually two-thirds the part thickness. 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 1](#) below illustrates the terms we use to describe gate size.

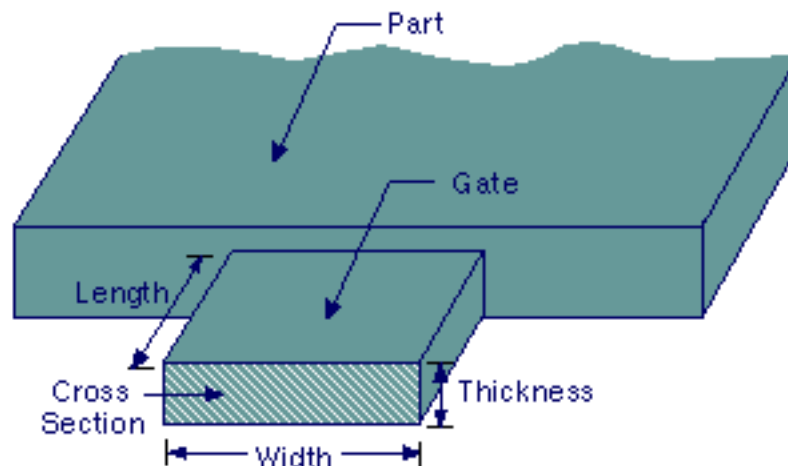


FIGURE 1. Gate size terminology

Gate location

Select a gate location that will ensure rapid and uniform mold filling. Position weld lines and air/gas vents so they have the least effect on the appearance and strength of the part. Since gates are locations of high residual stress, position them away from areas that will experience high external stress during use.

 **Position the gate away from load-bearing areas.**

The high melt pressure and high velocity of flowing material at a gate cause the area near a gate to be highly stressed.

 **Position the gate away from the thin section areas, or regions of sudden thickness change.**

This will avoid Hesitation or Sink marks and voids.

Gate types

Two categories of gates

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

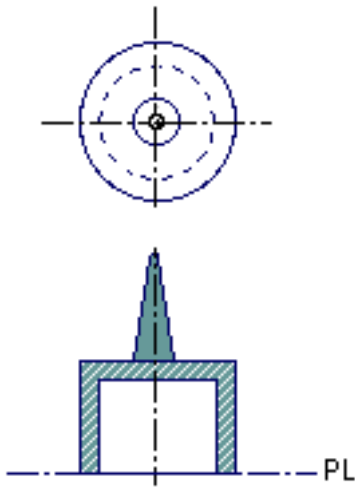
Manually trimmed gates

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 of molecules often precludes automatic gate trimming.

The following gate types are trimmed from the cavity manually:

Gate type
<u>Direct (sprue) gate</u>
<u>Tab gate</u>
<u>Edge (standard) gate</u>
<u>Overlap gate</u>
<u>Fan gate</u>
<u>Disk (diaphragm) gate</u>
<u>Ring gate</u>
<u>Spoke (spider) gate</u>
<u>Film (flash) gate</u>



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; 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 diameter here must be about 1.0 mm larger than the nozzle exit diameter. Standard sprue bushings have a taper of 2.4 degrees, opening toward the part. Therefore, the sprue length will control the diameter of the gate where it meets the part; the diameter should be at least 1.5 mm larger than or approximately twice the thickness of the part at that point.

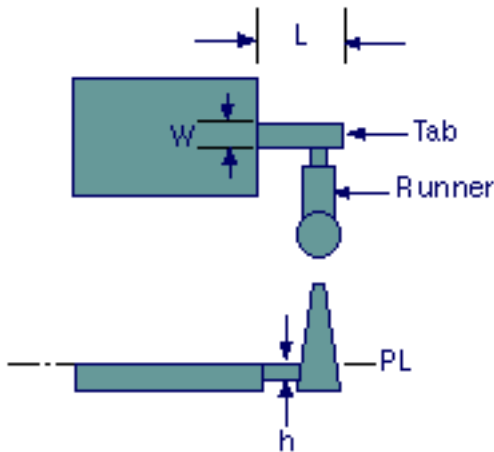
- A smaller taper angle (a minimum of one degree) risks not releasing the sprue from the sprue bushing on ejection.
- A larger taper wastes material and extends cooling time.
- Non-standard sprue tapers will be more expensive, with little gain.

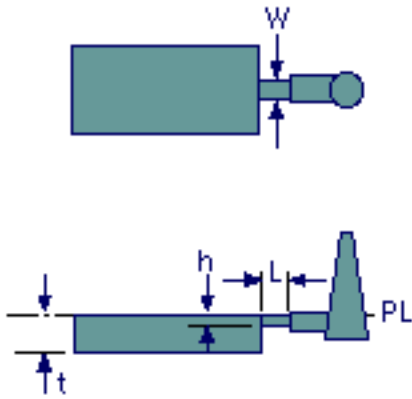
Tab gate

A tab gate is typically 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.



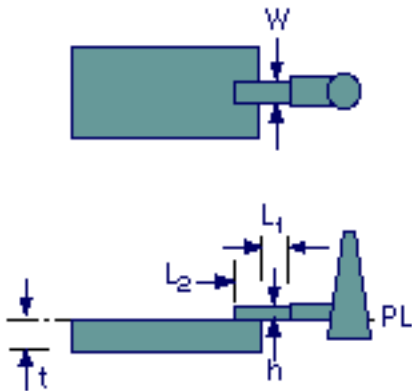


Edge (standard) gate

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 6% 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.

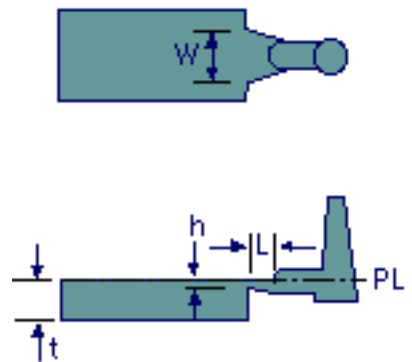


Overlap gate

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.



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.

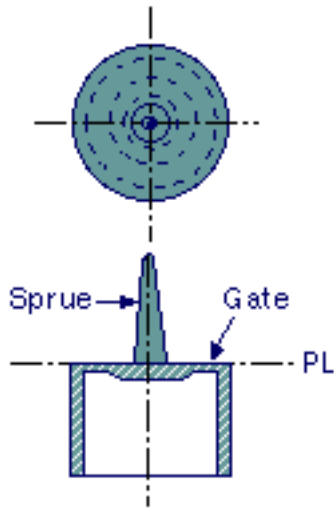
The gate should taper in both width and thickness, to maintain a constant cross sectional area. This will ensure that:

1. The melt velocity will be constant
2. The entire width is being used for the flow
3. The pressure is the same across the entire width.

► Dimensions

As with other manually trimmed gates, the maximum thickness should be no more than 75% of the part thickness.

Typical gate sizes are from 0.25 to 1.6 mm thick. The gate width is typically from 6.4 mm to 25% of the cavity length.



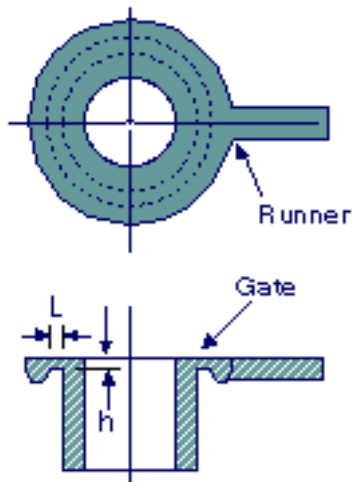
Disk (diaphragm) gate

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.

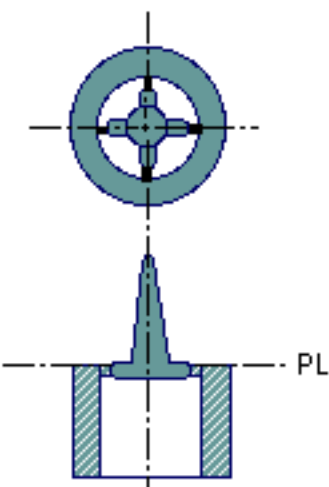


Ring gate

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.

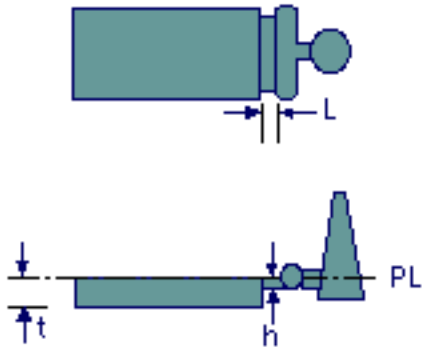


Spoke (spider) gate

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 the possibility of weld lines and the fact that 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.



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.

► 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.

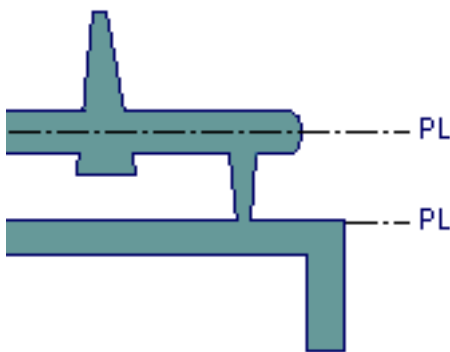
Automatically trimmed gates

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.

The following gate types are trimmed from the cavity automatically:

- Pin gate
- Submarine (tunnel, chisel) gate
- Hot-runner (hot-probe) gate
- Valve gate



Pin gate

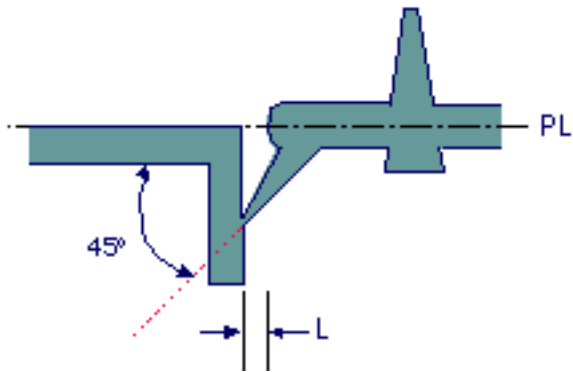
This type of gate relies on a three-plate mold design, where the runner system is on one mold parting line and the part cavity is in the primary parting line. Reverse taper runners drop through the middle (third) plate, parallel to the direction of the mold opening. As the mold cavity parting line is 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.

► 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.



Submarine (tunnel, chisel) gate

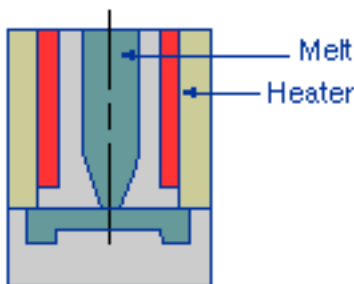
A submarine gate is used in two-plate mold construction. An angled, tapered tunnel is machined from the end of the runner to the cavity, just below the parting line. As the parts and runners are ejected, the gate is sheared at the part.

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

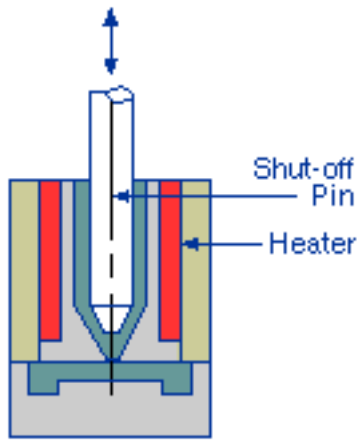
The typical size is 0.25 to 2.0 mm in diameter. It is tapered to the spherical side of the runner.



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, producing runnerless moldings.

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.



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 smooths 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.

Design rules

Essentials of a well-designed gate

A properly designed gate will deliver a rapid, uniform, and preferably uni-directional mold filling pattern with a proper gate freeze-off time.

Gate location

The gate location should be at the thickest area of the part, preferably at a spot where the function and appearance of the part are not impaired. This leads the material to flow from the thickest areas to thinner areas to the thinnest areas, and helps maintain the flow and packing paths. Gate location should be central so that flow lengths are equal to each extremity of the part.

Numerical simulation of the molding process is an effective tool that can be used to compare the effects of various gate designs.

Avoiding common problems

Improperly positioned gates often cause the following problems; keep them in mind when designing the delivery system.

▶ Gate symmetrically to avoid warpage

Symmetrical parts should be gated symmetrically, to maintain that symmetry. Asymmetric flow paths will allow some areas to be filled, packed, and frozen before other areas are filled. This will result in differential shrinkage and probable warpage of the parts.

▶ Vent properly to prevent air traps

The gate location should allow the air present in the cavity to escape during injection to prevent [air traps](#). 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.

▶ Enlarge the gate to avoid jetting

Gate location and size should prevent [jetting](#), the string appearance or spaghetti-size strands of melt in short shots. 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.

▶ 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.

Gate length

Gate length should be as short as possible to reduce an excessive pressure drop across the gate. A suitable gate length ranges from 1 to 1.5 mm (0.04 to 0.06 inches).

Gate size

Gates should always be small at the beginning of the design process so they can be enlarged, if necessary. Reducing the gate size is not as easy as enlarging it.

Gate thickness

The gate thickness is normally 50 to 80 percent 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 percent of the gated wall section thickness, to avoid part distortion during gate breaking. Typical diameters at the gate end for pin and submarine gates range from 0.25 to 2.0 mm (0.01 to 0.08 inches).

Freeze-off time

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

Fiber-filled materials

Fiber-filled materials require larger gates to minimize breakage of the fibers when they pass through the gate. Using small gates such as submarine, tunnel, or pin gates can damage the fillers in filled materials. Gates that deliver a uniform filling pattern (such as an edge gate) and thus, a uniform fiber orientation distribution are preferable to point-type gates.

Design Example: Optimizing gates and ram speed profile

Purpose of this example

This example illustrates various aspects of the power of CAE software for the plastics industry, and how such software can be strategically applied. It shows how to put into practice many of the concepts that are discussed in detail in the Design and Processing > Physics topics. These topics include injection pressure, filling pattern, melt-front area and melt-front velocity, runner design and balancing, and gate design.

Description of this example

The example steps through the design considerations for a molded plastic part in a concurrent engineering environment. It illustrates the role of computer simulation at each iterative step in the process. The software package used for this example is C-MOLD Process Solution. Phase I uses C-MOLD Filling EZ to analyze the initial design and determine the gate location and ram speed profile; Phase II uses C-MOLD Filling and Post-Filling for a more detailed analysis of the material selection, part and tool design, and process conditions.

Phase I: Fast and easy simulation with C-MOLD Filling EZ

In this first phase of the design process, we run an initial simulation using C-MOLD Filling EZ, which performs a three-dimensional mold-filling simulation of Newtonian fluids under isothermal conditions. Filling EZ is designed for preliminary analysis of part, gate, and process designs. We then go through several design iterations, each time modifying the design variables (such as location and number of gates), rerunning the analysis, and observing the results. At the end of Phase I, we will have determined the best location for the gate and an optimal ram-speed profile.

► Given

An initial product design.

► Objective

To determine the proper gate location and optimal ram-speed profile.

► Design criteria

- Produce a uniform filling pattern with a minimum flow length and a minimum injection pressure requirement.
- Maintain a constant melt-front velocity to minimize property variation.
- Design for manufacturing and assembly, concurrent engineering, and early vendor involvement.

Design 1

Design 1 set-up

Gate: Fan gate at one side

Ram-speed profile:
Constant

Action taken

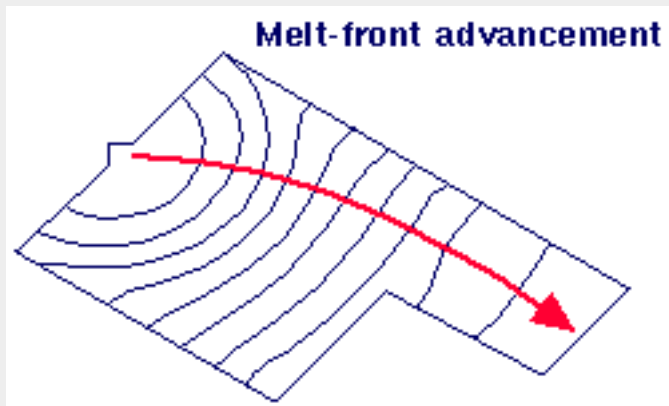
We ran the Filling EZ analysis with the variables specified in the Design 1 set-up.

Result 1.1

The melt-front advancement plot at the right shows that the maximum flow length is about the same as the part length.

Problem to overcome

A longer flow length generally requires a high injection pressure to fill the part.



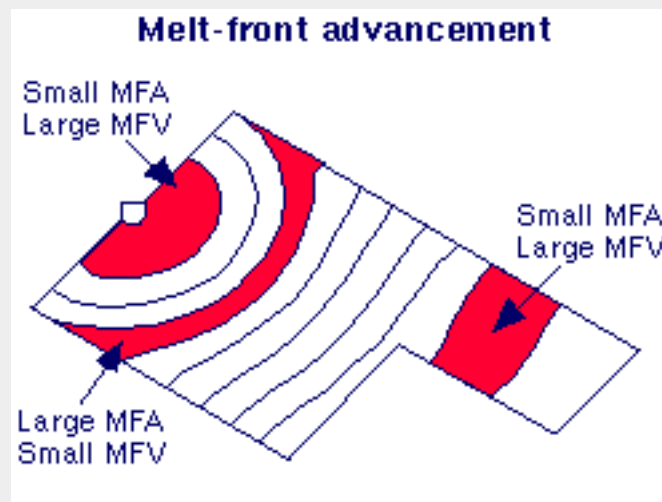
Long flow length requires high pressure

Result 1.2

With a constant ram speed (i.e., constant volumetric flow rate), the melt-front velocity varies with time, depending on the melt-front area.

Problem to overcome

A variable MFV introduces differential molecular or fiber orientation on the part surface region, potentially resulting



Material at melt front flows at different velocities

in differential shrinkage and warpage.

Design 2

Design 2 set-up

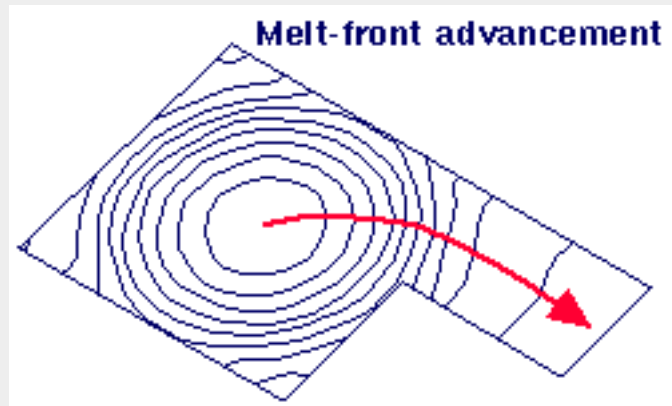
Gate: Center gate
Ram-speed profile: Constant

Action taken

We moved the gate to be near the geometric center of the part, and re-ran the analysis.

Result 2.1

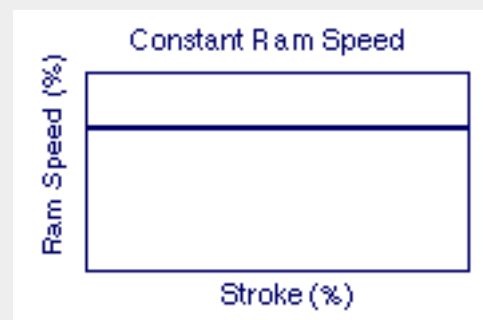
The maximum flow length is reduced, compared with [Design 1](#). This gate location change reduces the maximum flow length and thus the injection pressure.



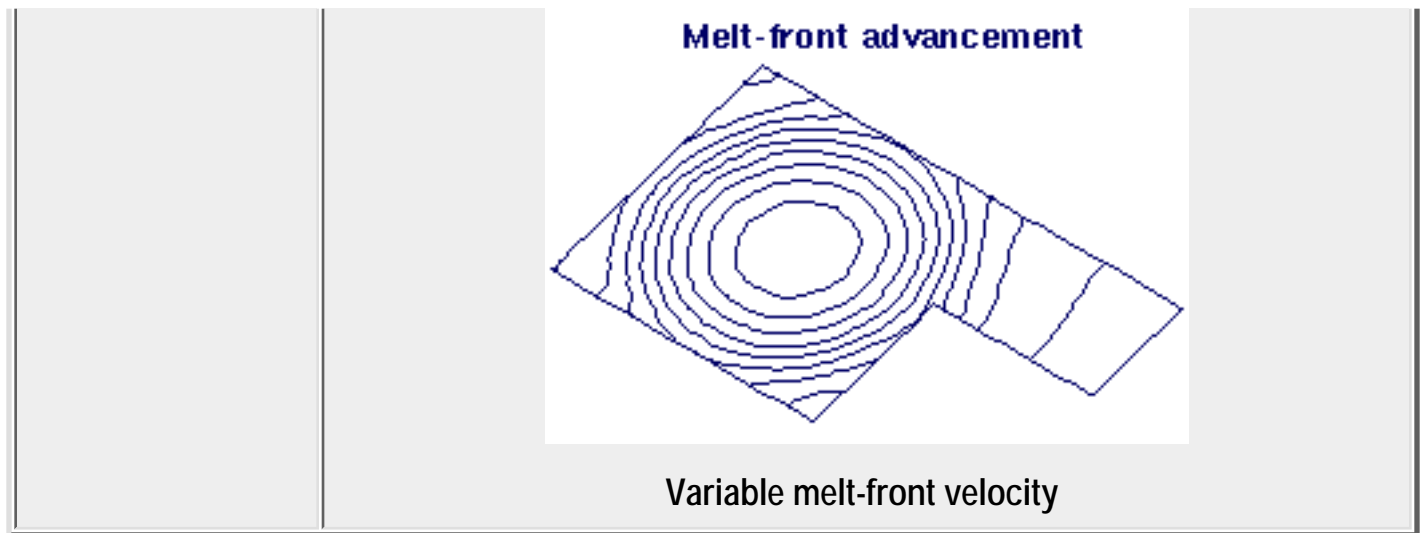
Flow length reduces with center gate

Problem to overcome

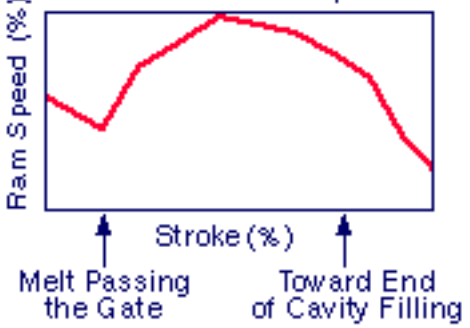
The initial radial filling pattern with a variable MFV remains undesirable.

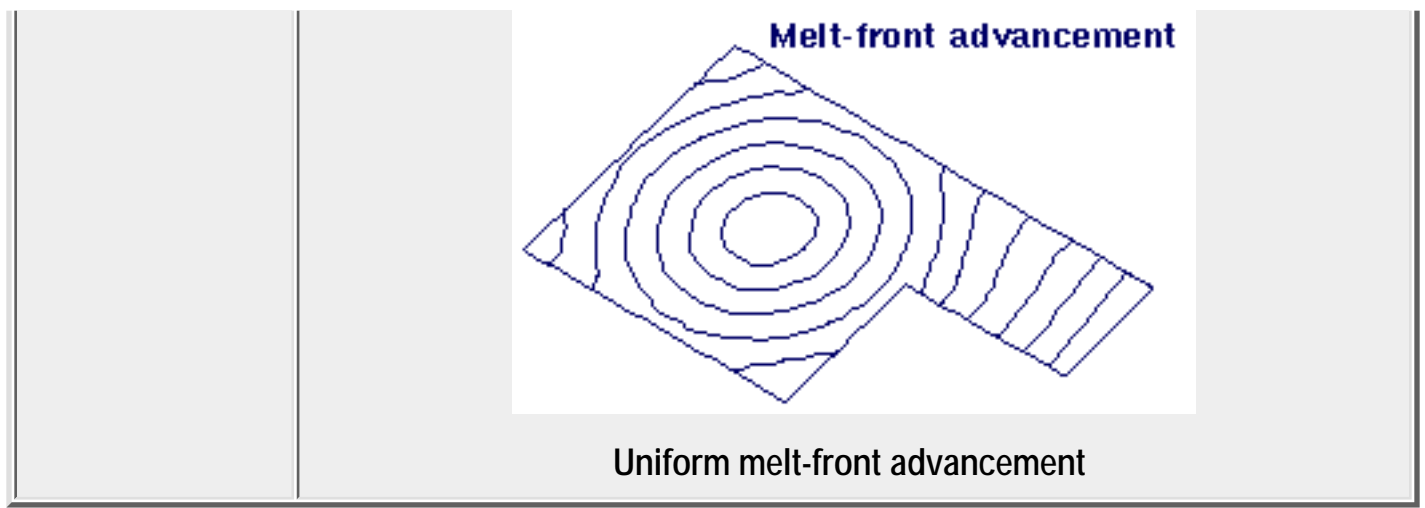


Constant ram-speed profile



Design 3

<p>Design 3 set-up Gate: Center gate Ram-speed profile: Variable</p>	<p>Action taken We used the optimal ram-speed profile recommended by Filling EZ results for Design 2 and re-ran the analysis.</p>
<p>Result 3.1 The variable MFV is eliminated by employing an optimal ram-speed profile.</p>	<div data-bbox="708 1157 1192 1535"> <p style="text-align: center;">Profiled Ram Speed</p>  <p style="text-align: center;">Variable ram-speed profile</p> </div>



Phase II: Optimize the design with C-MOLD Filling & Post Filling

The preliminary phase of the design process has produced some improvements on the initial design. The next phase employs C-MOLD Filling and Post-Filling analysis for a more detailed study. This is a three-dimensional mold-filling simulation that takes into account the heat transfer and non-Newtonian flow behavior of thermoplastics. It provides a more rigorous and detailed analysis, including examination of material selection, part and tool design, and process conditions. By the end of Phase II, we have fine-tuned the gate location and runner system, and have optimized the process conditions (such as fill time, and melt and coolant temperatures).

► Given

A product design with a preliminary gate position (from [Design 2](#)) and an optimal ram-speed profile (from [Design 3](#)).

► Objectives

To optimize the gate location, runner system, and process conditions for cavity filling.

► Design criteria

- Produce a uniform filling pattern to reduce the injection pressure requirement and clamp force requirement, to save energy and machine cost.
- Maintain a constant melt-front velocity to minimize property variation.
- Minimize spatial melt temperature variation to eliminate localized hot/cold spots and the resulting surface defects.
- Keep the maximum shear stress and shear rate levels within the recommended value given by the resin supplier.

Design 4

Design 4 set-up

Gate location:

Center

Ram-speed profile:

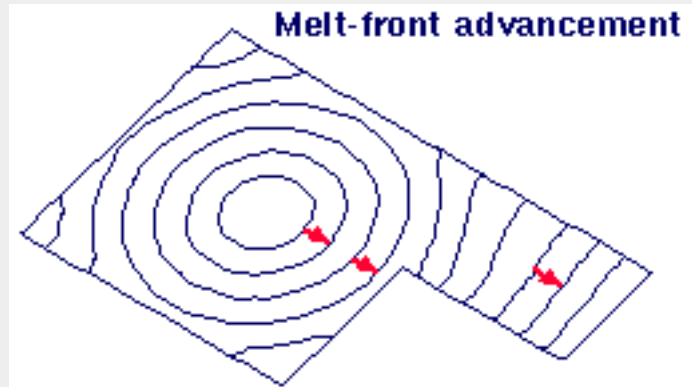
Variable

Action taken

We ran a series **Filling & Post-Filling** analyses with different feasible fill times (i.e., a fill-time scan).

Result 4.1

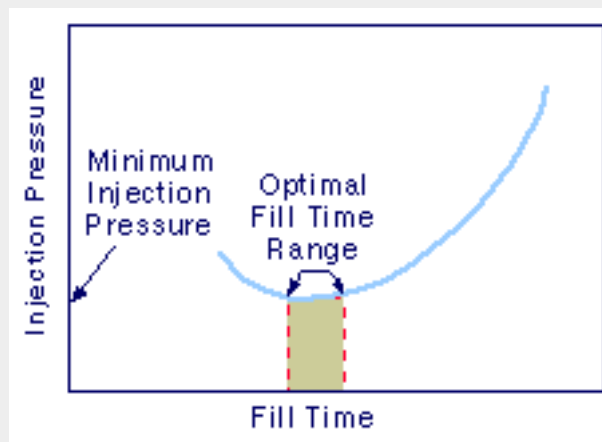
The melt-front advancements show a constant space between the contours, an indication of constant MFV.



Constant melt-front velocity

Result 4.2

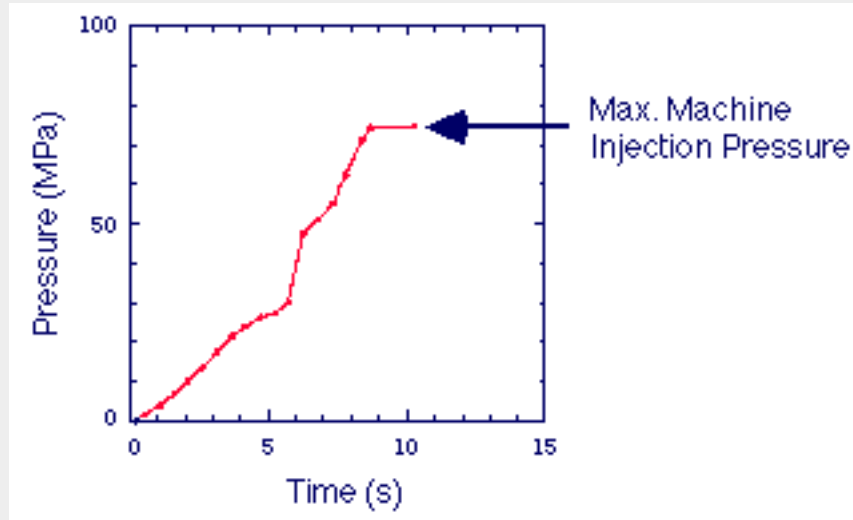
The fill-time scan we ran established a U-shaped process curve. When plotting the required injection pressure vs. various fill times, we can find the optimal fill-time range that requires the least injection pressure.



Optimal fill time

What if?

If the required injection pressure exceeds the maximum machine capability (80 MPa in this case), the process conditions or design must be modified.



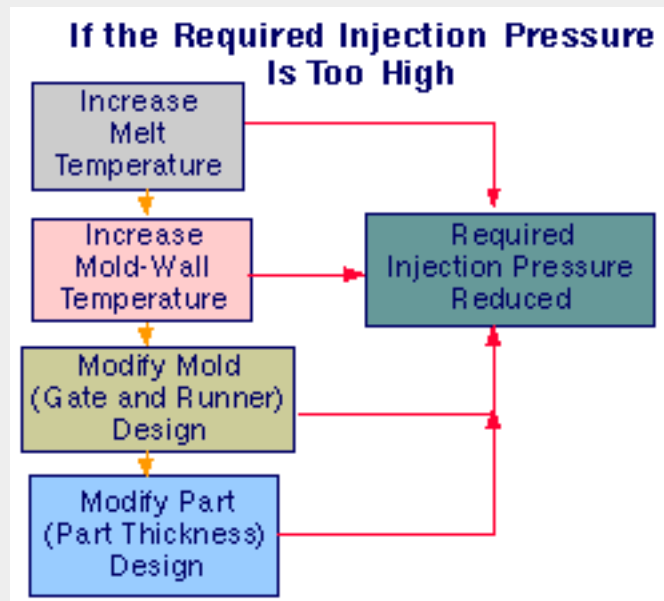
Injection pressure too high

Solution

■ Always start with simple and less expensive remedies that use existing tools, and evaluate the pros and cons of each option.

■ Analyze alternative cases with different melt and mold-wall temperatures, gate and runner designs, or part thickness.

■ Iterate all competing designs and select the optimal one.



Try alternative process conditions

Design 5

Design 5 set-up

Gate location:

Multiple

Ram-speed profile:

Variable

Action taken

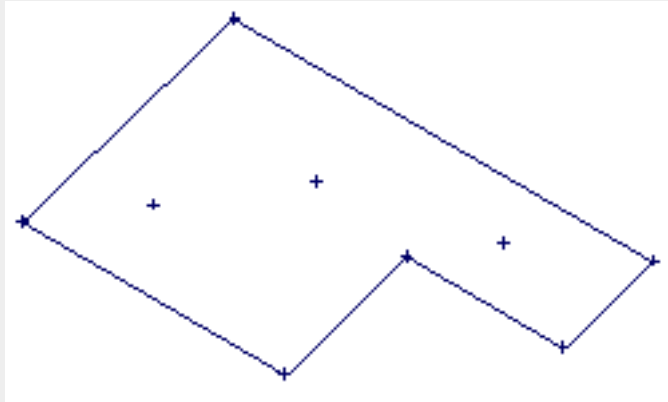
We ran a Filling EZ analysis to determine the proper locations for multiple gates and an optimal ram-speed profile.

Solution

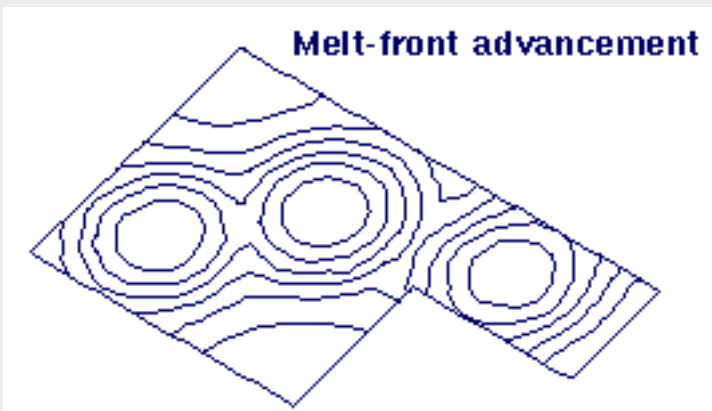
Since it is in the early design phase, we decided to modify the tool design instead of changing the melt/mold temperatures, which may adversely increase the cycle time. In the design at the right, we employed additional gates to reduce the flow length. This reduced the injection pressure requirement.

Result 5.1

Filling EZ predicted the new filling pattern with gates at three possible locations.



Selected gate locations



Melt-front advancement with the modified gate design

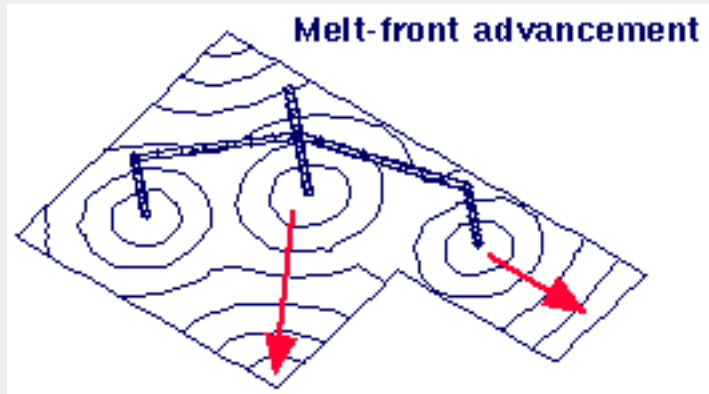
Design 6

Design 6 set-up**Gates: Three gates****Runners:****Hot-runner system****Ram-speed profile:****Variable****Action taken****We added a hot-runner system and ran a Filling & Post-Filling analysis.****Result 6.1**

With multiple gates, the flow length was significantly reduced, resulting in a lower injection pressure requirement.

Result 6.2

Adding a hot-runner system reduced the scrap from runners with minimum pressure drop within the runners.



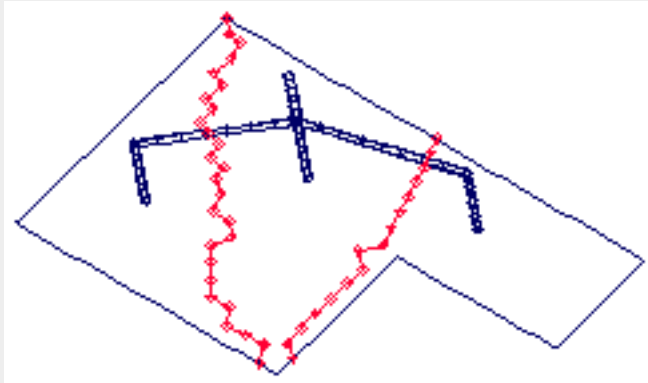
Lower pressure with shorter flow length

Problem to overcome

One major drawback of the multi-gate system is the formation of weld and meld lines. C-MOLD Filling analysis can predict their location.

Suggestion

If the location of weld lines is unacceptable, you can change the gate location or program the opening and closing of valve gates to alter the



Multiple gates lead to weld lines

DESIGN 7

Design 7 set-up

Gates: Three valve gates

Runners:

Hot-runner system

Ram-speed profile:

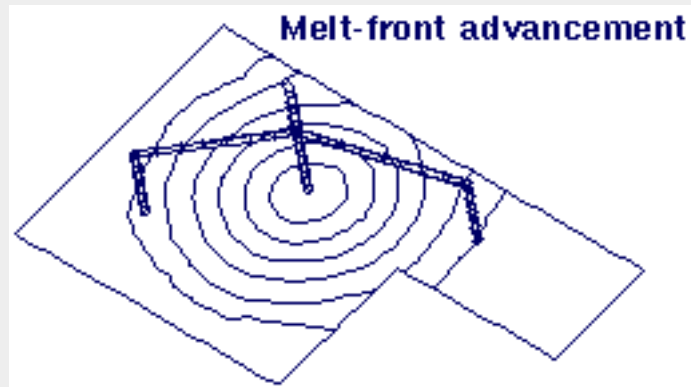
Variable

Action taken

We added valve gates for sequential opening and closing control.

Result 7.1

Using sequential valve-gate openings eliminated the weld line while keeping the merits of multiple gates.

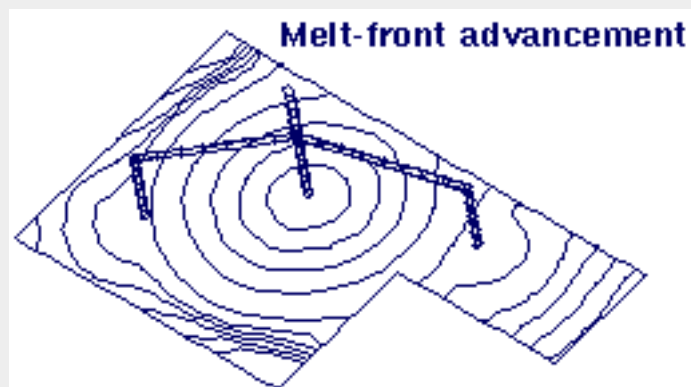


Initially, only center valve gate is opened

Result 7.2

The gates downstream remain closed until the melt front arrives, when they will open to continue the filling process.

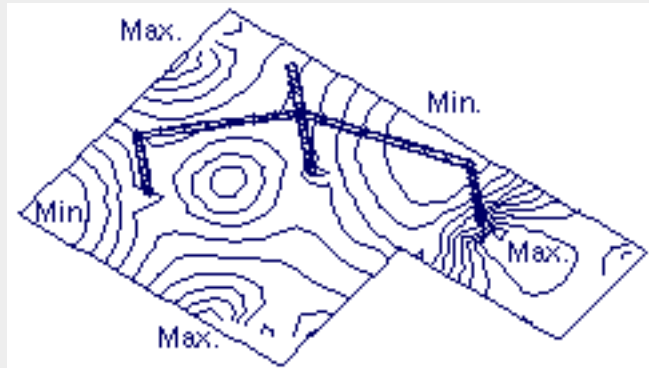
The center gate can either be closed or remain open after the other downstream gates open.



Downstream gate opens after melt arrives

Result 7.3

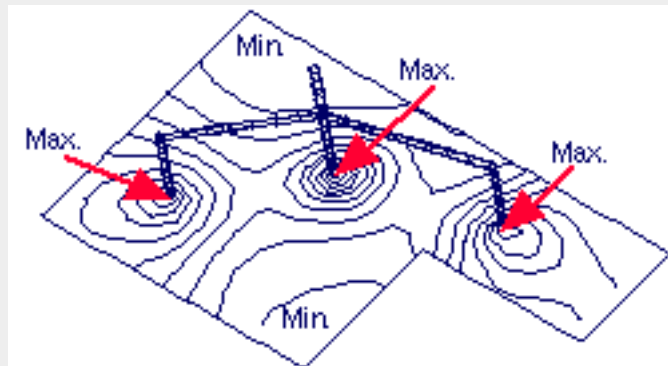
The bulk temperature distribution shown by C-MOLD gives the spatial temperature variation across the part at various time instants during filling and post-filling stages.



Bulk-temperature distribution

Result 7.4

Melt fracture or dimensional instability will occur when the maximum shear stress exceeds a certain level. The wall-shear stress distribution shown by C-MOLD can be used to pinpoint the potential problem.



Wall-shear stress distribution

Mold cooling system overview

Importance of cooling system design

Mold cooling accounts for more than two-thirds of the total cycle time in the production of injection molded thermoplastic parts. [Figure 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 2](#)).

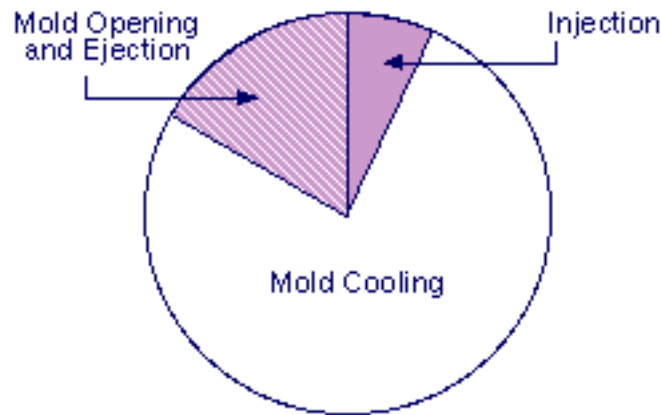


FIGURE 1. Mold cooling accounts for more than two-thirds of the total cycle time

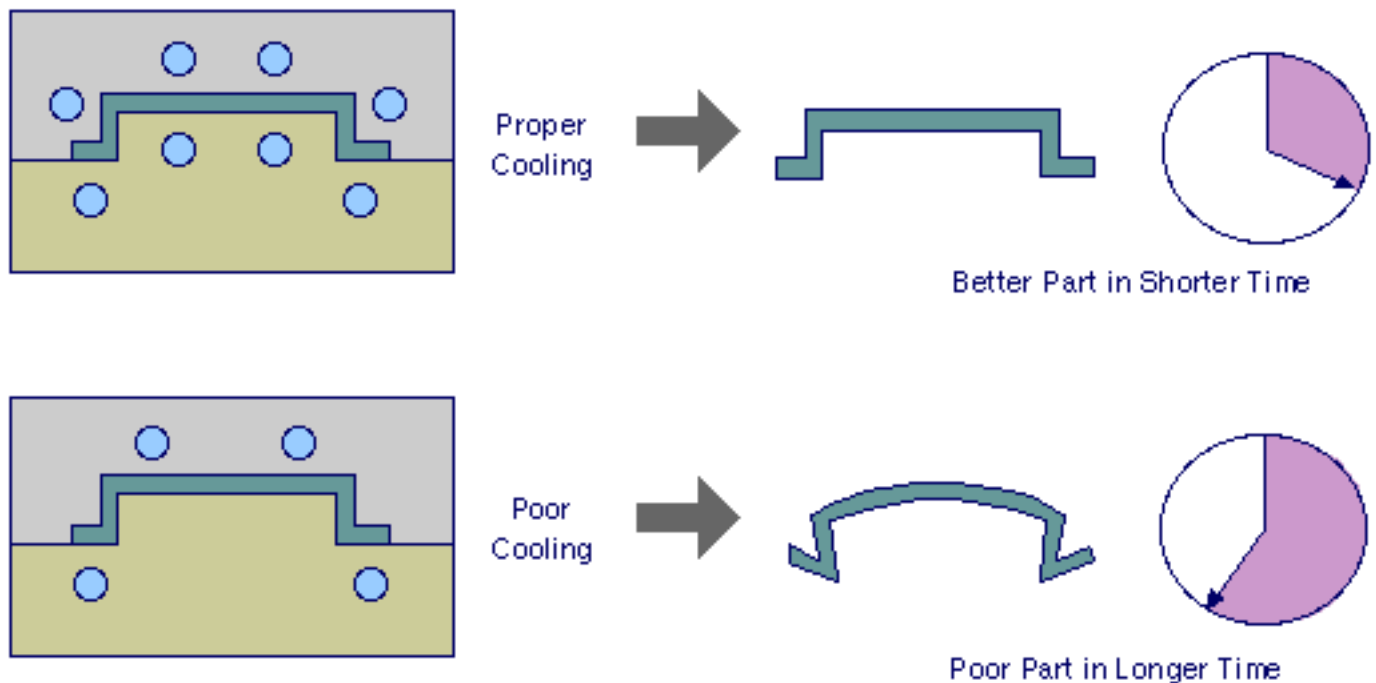


FIGURE 2. Proper and efficient cooling improves part quality and productivity

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.

Figures [3](#) and [4](#) illustrate the components of a typical cooling system.

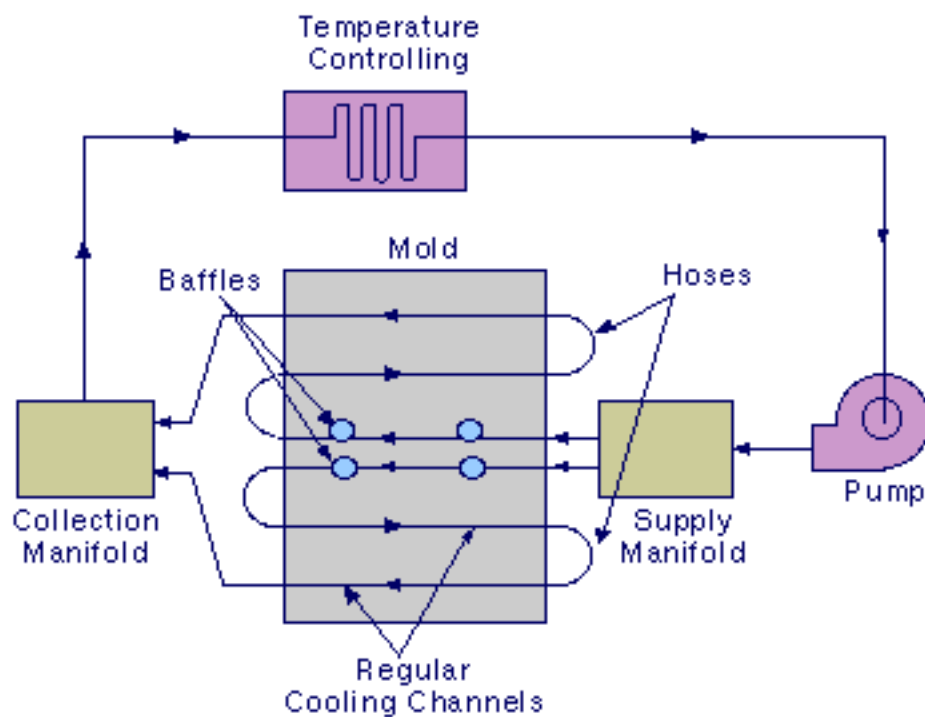


FIGURE 3. A typical cooling system for an injection molding machine.

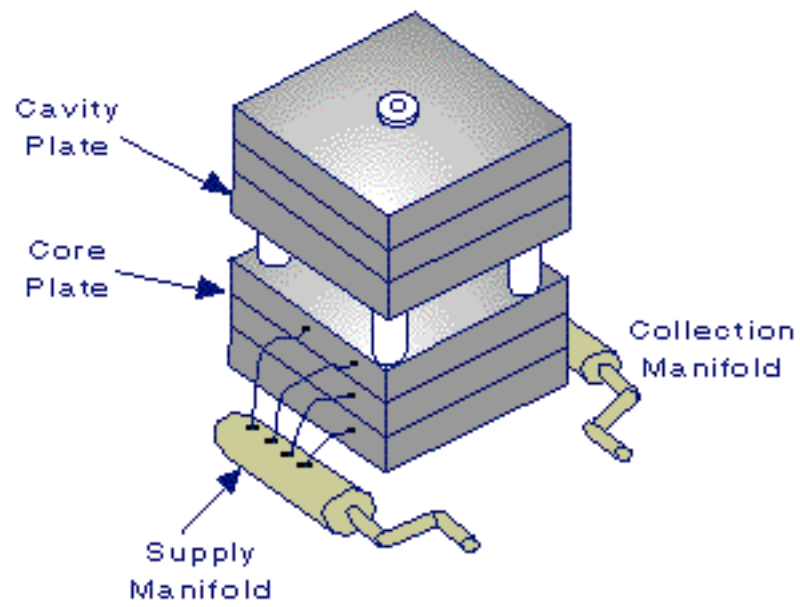


FIGURE 4. A cooling channel assembly attached to the mold plates.

Cooling-channel configuration

Types of cooling channels

Cooling-channel configurations can be serial or parallel. Both configurations are illustrated in [Figure 1](#) below.

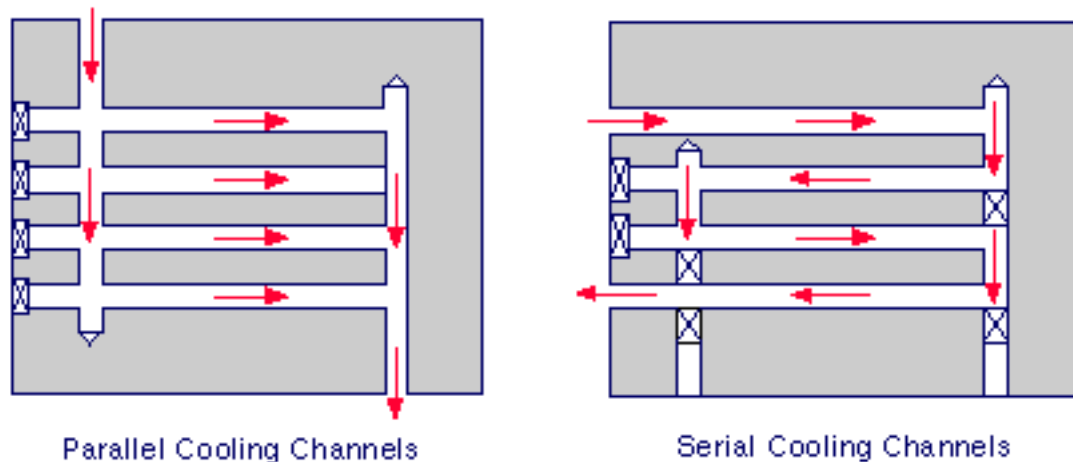


FIGURE 1. Cooling-channel configurations

Parallel cooling channels

Parallel cooling channels are drilled straight through from a supply manifold to a collection manifold. Due to the flow characteristics of the parallel design, the flow rate along various cooling channels may be different, depending on the flow resistance of each individual cooling channel. 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 may not be uniform with a parallel cooling-channel configuration.

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.

Serial cooling channels

Cooling channels connected in a single loop from the coolant inlet to its outlet are called serial 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](#) discusses this more thoroughly. However, you should take care to minimize the temperature rise of the coolant, since the coolant will collect all the heat along the entire cooling-channel path. In general, the temperature difference of the coolant at the inlet and the exit should be within 5°C for general-purpose molds and 3°C for precision molds. For large molds, more than one serial cooling channel may be required to assure

uniform coolant temperature and thus uniform mold cooling.

Alternative cooling devices

What do they do?

Baffles and bubblers are sections of cooling lines that divert the coolant flow into areas that would normally lack cooling. Cooling channels are typically drilled through the mold cavity and core. The mold, however, may consist of areas too far away to accommodate 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 below.

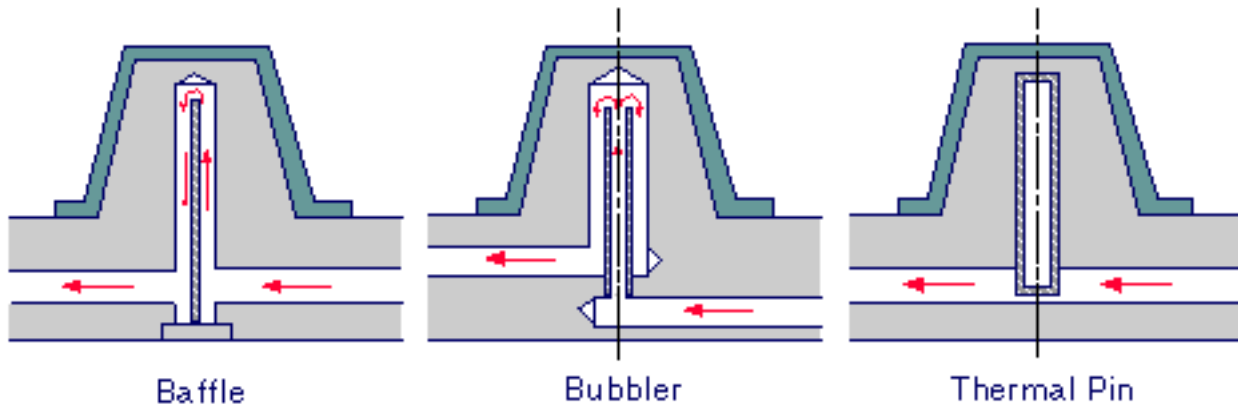


FIGURE 1. Baffle, bubbler, and thermal pin

Baffles

A baffle is actually a cooling channel drilled perpendicular to a main cooling line, with a blade that separates one cooling passage into two semi-circular 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, then flows back to the main cooling line.

This method provides maximum cross sections for the coolant, but it is difficult to mount the divider exactly in the center. The cooling effect and with it the temperature distribution on one side of the core may differ from that on the other side. This disadvantage of an otherwise economical solution, as far as manufacturing is concerned, can be eliminated if the metal sheet forming the baffle is twisted. For example, the helix baffle, as shown in [Figure 2](#) below, 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 2](#) below.

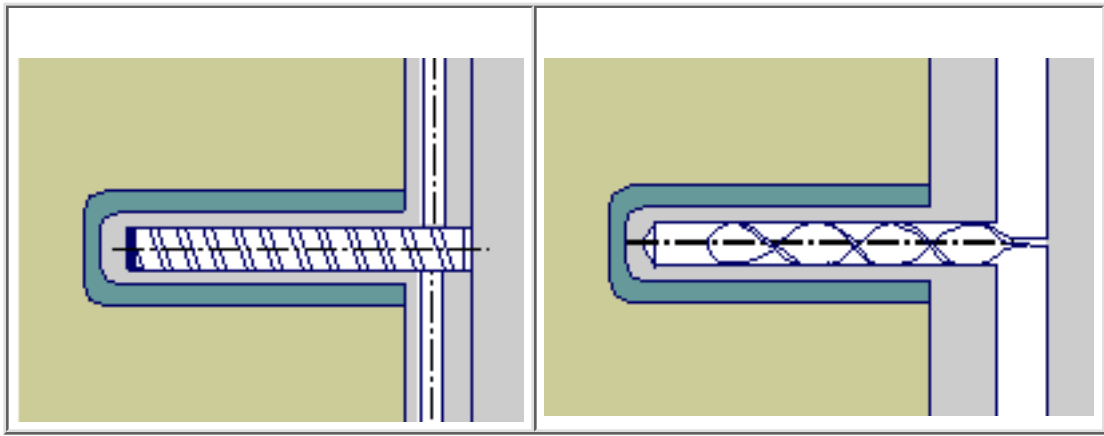


FIGURE 2. (Left) Helix baffle. (Right) Spiral baffle.

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:

$$\text{Inner Diameter} / \text{Outer Diameter} = 0.707$$

Bubblers are commercially available and are usually screwed into the core, as shown in [Figure 3](#) below. Up to a diameter of 4 mm, the tubing should be beveled at the end to enlarge the cross section of the outlet; this technique is illustrated in [Figure 3](#). Bubblers can be used not only for core cooling, but are also for cooling flat mold sections, which can't be equipped with drilled or milled channels.

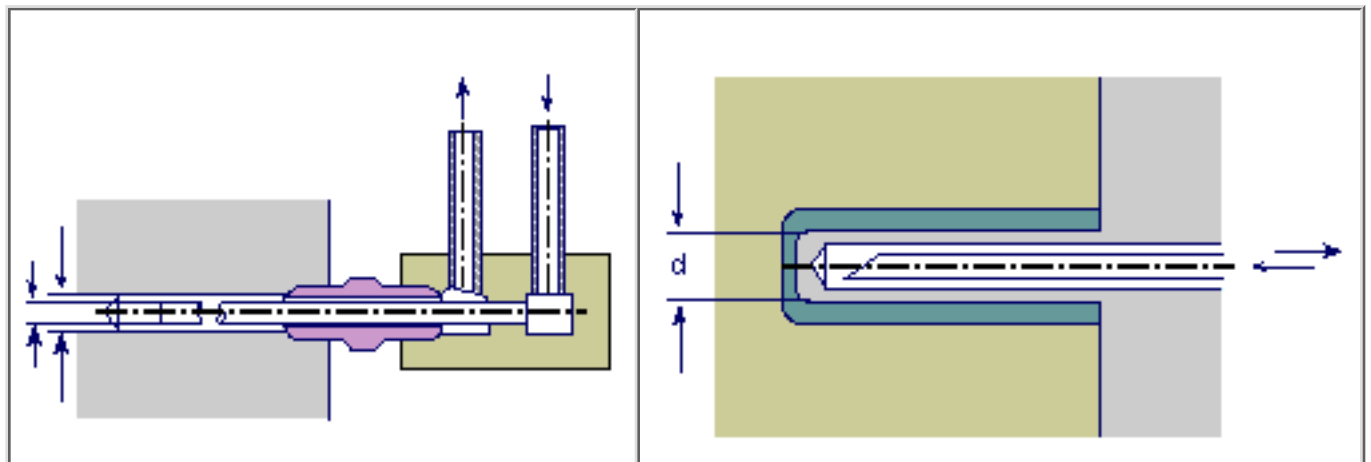


FIGURE 3. (Left) Bubblers screwed into core. (Right) Bubbler beveled to enlarge outlet

NOTE: Because both baffles and bubblers 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 C-MOLD Cooling analysis.

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 4](#). The heat transfer efficiency of a thermal pin is almost ten 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.

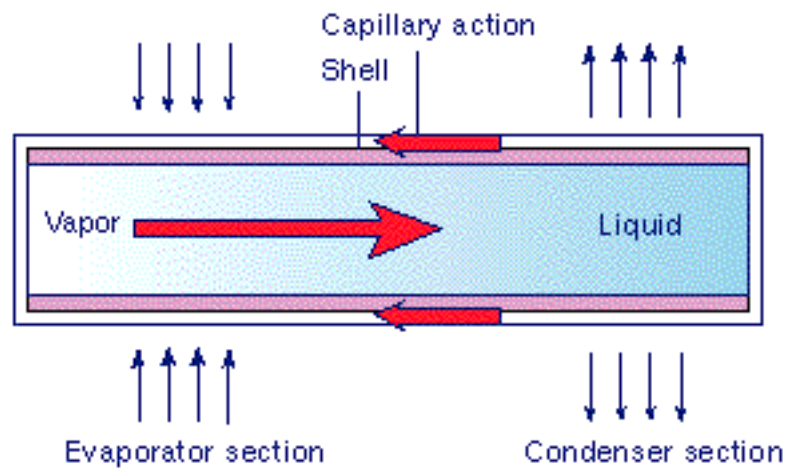


FIGURE 4. Thermal pin heat transfer efficiency

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 5](#). This procedure, of course, does not permit maintaining an exact mold temperature.

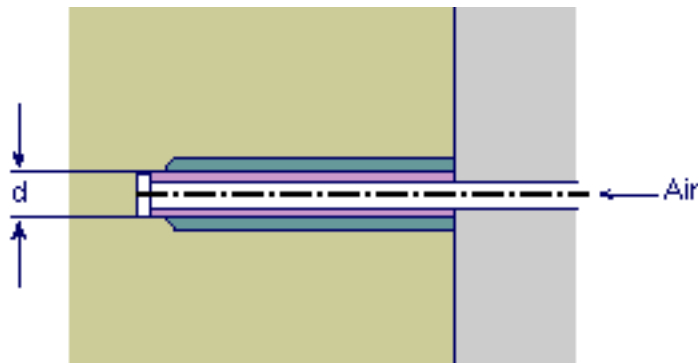
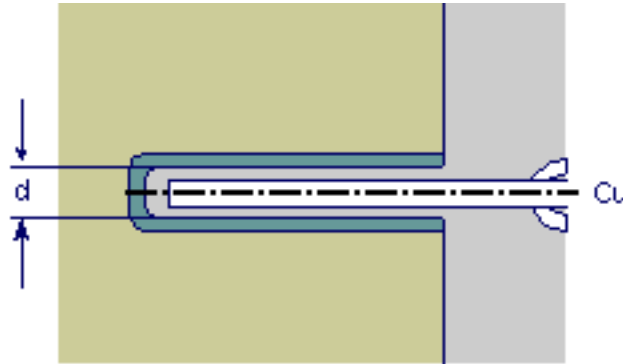


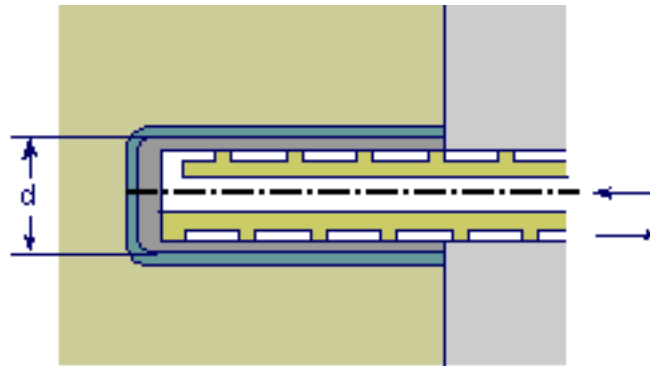
FIGURE 5. Air cooling of a slender core

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 or beryllium-copper materials. This technique is illustrated in [Figure 6](#) below. Such inserts are press-fitted into the core and extend with their base, which has a cross section as large as is feasible, into a cooling channel.

**FIGURE 6.** Using high thermal conductivity material to cool a slender core

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 7](#). This design weakens the core significantly.

**FIGURE 7.** Use of helical baffle to cool large core

Cooling cylinder cores

Cooling of cylinder cores and other round parts should be done with a double helix, as shown below. 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.

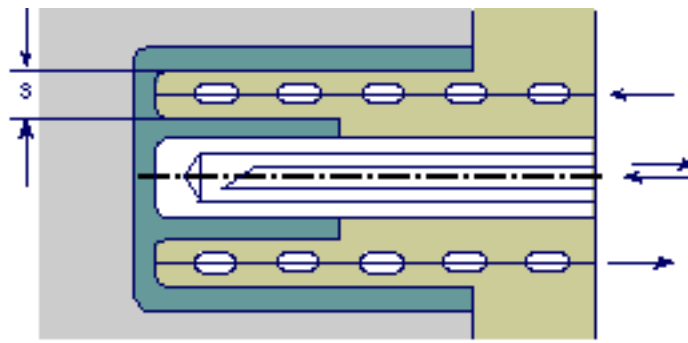


FIGURE 8. Double helix with center bubbler

Cooling system equations

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:

$$\text{Cooling time} \propto \frac{(\text{Heaviest wall thickness})^2}{(\text{thermal diffusivity of polymer melt})}$$

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

where the thermal diffusivity of polymer melt is defined as

$$\text{Thermal diffusivity} = \frac{(\text{Thermal conductivity})}{(\text{Density}) (\text{Specific heat})}$$

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

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 1](#). The Reynolds number is defined as:

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

where ρ 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 1. Coolant flow types and corresponding Reynolds number ranges

Reynolds Number (Re)	Type of Flow
10,000 < Re	Turbulent Flow

$2,300 < Re < 10,000$	Transition Flow
$100 < Re < 2,300$	Laminar Flow
$Re < 100$	Stagnated Flow

Residual stress

A culprit in shrinkage and warpage problems

Residual stress is a process-induced stress, frozen in a molded part. It can be either **flow-induced** or **thermal-induced**. Residual stresses affect a part similarly to externally applied stresses. If they are strong enough to overcome the structural integrity of the part, the part will warp upon ejection, or later crack, when external service load is applied. Residual stresses are the main cause of part shrinkage and warpage. The process conditions and design elements that reduce shear stress during cavity filling will help to reduce flow-induced residual stress. Likewise, those that promote sufficient packing and uniform mold cooling will reduce thermal-induced residual stress. For fiber-filled materials, those process conditions that promote uniform mechanical properties will reduce thermal-induced residual stress.

Flow-induced residual stress

Unstressed, long-chain polymer molecules tend to conform to a random-coil state of equilibrium at temperatures higher than the melt temperature (i.e., in a molten state). During processing the molecules orient in the direction of flow, as the polymer is sheared and elongated. If solidification occurs before the polymer molecules are fully relaxed to their state of equilibrium, molecular orientation is locked within the molded part. This type of frozen-in stressed state is often referred to as *flow-induced residual stress*. Because of the stretched molecular orientation in the direction of flow, it introduces anisotropic, non-uniform shrinkage and mechanical properties in the directions parallel and perpendicular to the direction of flow.

Frozen-in molecular orientation

Due to a combination of high shear stress and a high cooling rate adjacent to the mold wall, there is a highly oriented layer frozen immediately below the part surface. This is illustrated in [Figure 1](#). Subsequent exposure of a part with high residual flow stresses (or frozen-in orientation) to high temperature may allow some of the stresses to relieve. This typically results in part shrinkage and warpage. Due to the thermal insulating effect of the frozen layers, polymer melt in the hot core is able to relax to a higher degree, leading to a low molecular orientation zone.

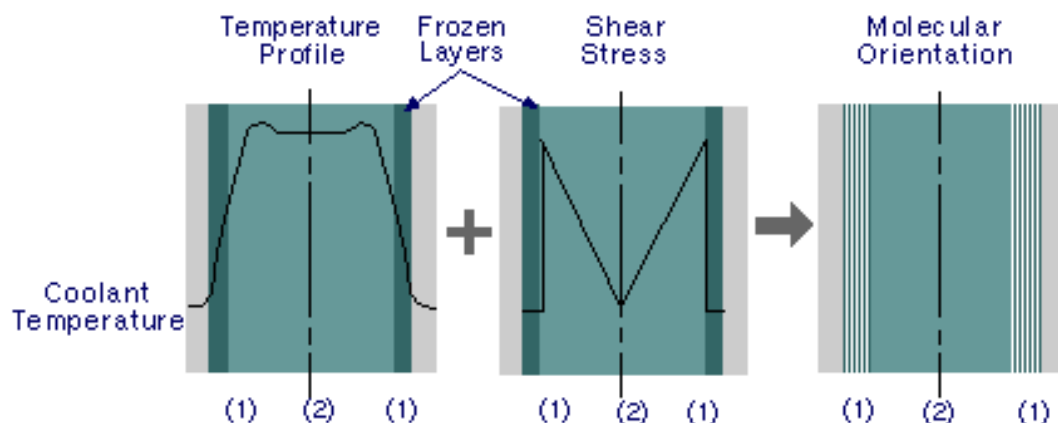


FIGURE 1. The development of residual flow stresses due to frozen-in molecular orientation during the filling and packing stages. (1) High cooling, shear, and orientation zone (2) Low cooling, shear, and orientation zone

Reducing flow-induced residual stress

Process conditions that reduce the shear stress in the melt will reduce the level of flow-induced residual stresses. In general, flow-induced residual stress is one order of magnitude smaller than the thermal-induced residual stress.

- higher melt temperature
- higher mold-wall temperature
- longer fill time (lower melt velocity)
- decreased packing pressure
- shorter flow path.

Thermal-induced residual stress

Thermal-induced residual stress occurs due to the following reasons:

- Material shrinks as the temperature drops from the process settings to the ambient conditions reached when the process is complete.
- The material elements experience different thermal-mechanical histories (e.g., different cooling rates and packing pressures) as the material solidifies from the mold wall to the center.
- Changing pressure, temperature, and molecular and fiber orientation result in variable density and mechanical properties.
- Certain mold constraints prevent the molded part from shrinking in the planar directions.

Free quenching example

Material shrinkage during injection molding can be conveniently demonstrated with a *free quenching* example, in which a part of uniform temperature is suddenly sandwiched by cold mold walls. During early cooling stages, when the external surface layers cool and start to shrink, the bulk of the polymer at the hot core is still molten and free to contract. However, as the internal core cools, local thermal contraction is constrained by the already-rigid external layers. This results in a typical state of stress distribution with tension in the core balanced by compression in the outer layers, as illustrated [Figure 2](#) below.

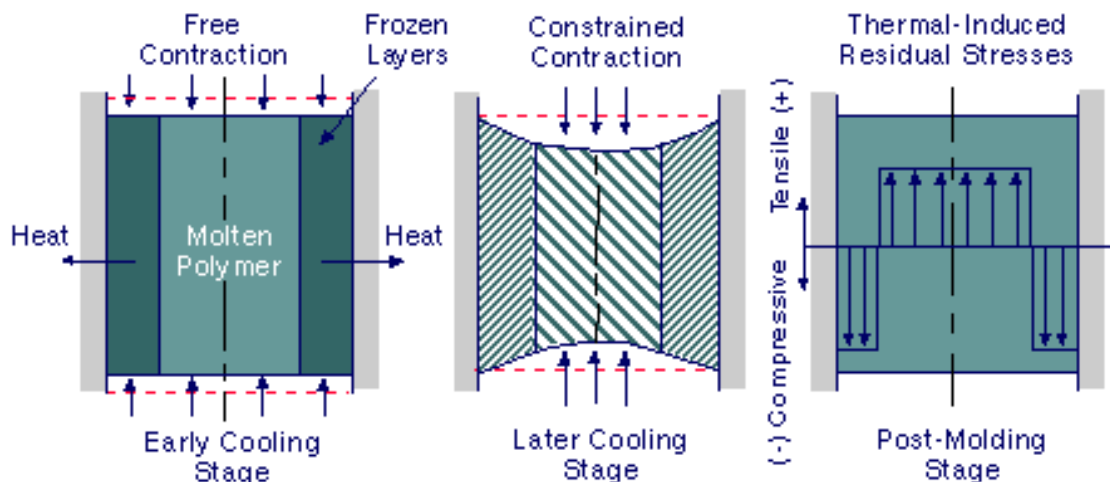


FIGURE 2. The development of residual thermal stress in a "free-quenching" part due to variations in cooling across the molded part and the material's response to the temperature history.

Unbalanced cooling

Variation in the cooling rate from the mold wall to its center can cause thermal-induced residual stress. Furthermore, asymmetrical thermal-induced residual stress can occur if the cooling rate of the two surfaces is unbalanced. Such unbalanced cooling will result in an asymmetric tension-compression pattern across the part, causing a bending moment that tends to cause part warpage. This is illustrated in [Figure 3](#) below. Consequently, parts with non-uniform thickness or poorly cooled areas are prone to unbalanced cooling, and thus to residual thermal stresses. For moderately complex parts, the thermal-induced residual stress distribution is further complicated by non-uniform wall thickness, mold cooling, and mold constraints to free contraction.

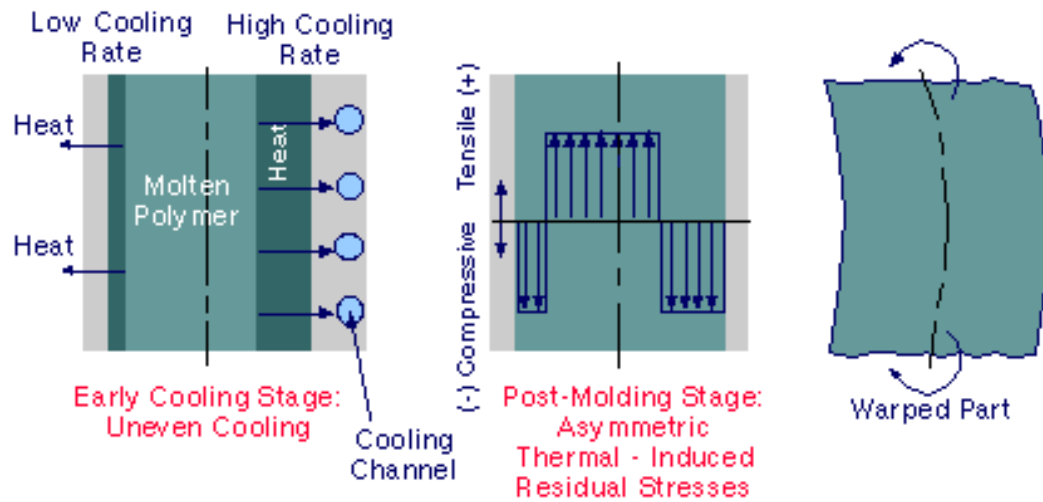


FIGURE 3. Asymmetrical thermal-induced residual stress caused by unbalanced cooling across the molded part thickness introduces part warpage

Variable frozen-in densities

The figure below illustrates the variation in frozen-in densities caused by the packing pressure history.

► Temperature profile

The left figure plots the temperature profile at one location on the part. For the purpose of illustration, the part is divided into eight equal layers across the part thickness. The profile shows the temperature at the solidification (freeze-off) time instant for each layer (t_1 to t_8). Note that the material starts solidifying from the outer layers and the frozen interface moves inwards with time.

► Pressure trace

The center figure plots a typical pressure history, showing the pressure levels (P_1 to P_8) as each layer solidifies. In general, the pressure gradually increases during filling, reaching a maximum in the early packing stage, and then starts to decay due to cooling and gate freeze-off. Accordingly, the material at the outer layers and center layers solidify when the pressure level is low, whereas the intermediate layers freeze under high packing pressure.

► Frozen-in specific volume

The right figure depicts the specific volume trace for layer 5 on a pVt plot and the final frozen-in specific volumes for all the layers, marked by the numbered solid circles.

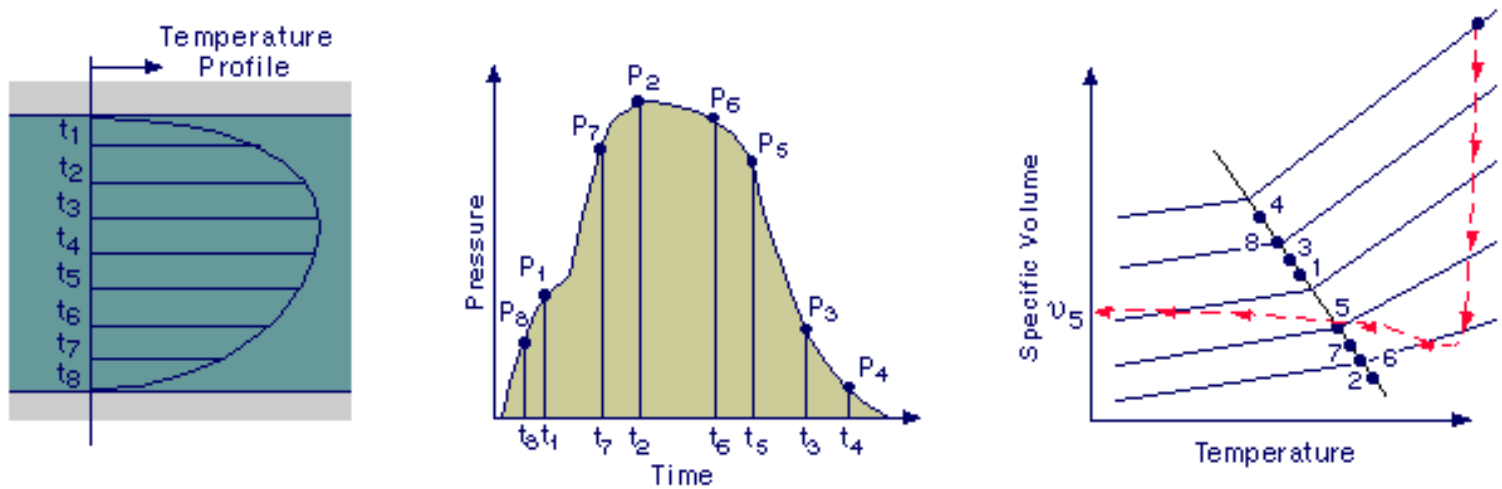


FIGURE 4. Factors that influence the development of "frozen-in" specific volume

Differential shrinkage

Given the frozen-in specific volumes, the various layers will shrink differently, according to the $p-v-T$ curves that govern the material shrinkage behavior. Hypothetically, if each layer were detached from others (as shown in [Figure 5](#)) then material elements in the left figure below would have shrunk like those in the center figure. In this case, the intermediate layers tend to shrink less than the others because of lower frozen-in specific volume (or, equivalently, higher frozen-in density). In reality, all the layers are bound together. Therefore, the end result will be a compromised shrinkage distribution with intermediate layers being compressed and outer and center layers being stretched.

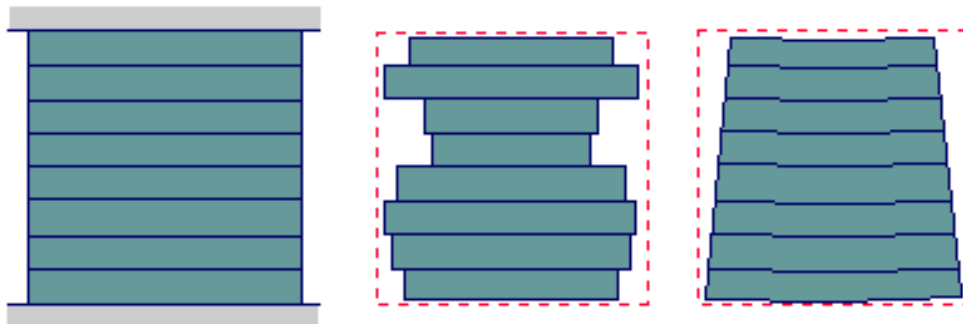


FIGURE 5. Variable residual stresses arise and the part deforms as layers of different frozen-in specific volume interact with each other

Process-induced vs. in-cavity residual stress

Process-induced residual stress data are much more useful than in-cavity residual stress data for molding simulation. Following are definitions of the two terms, along with an example that illustrates the difference between them.

Process-induced residual stress

After part ejection, the constraints from the mold cavity are released, and the part is free to shrink and deform. After it settles to an equilibrium state, the remaining stress inside the part is called process-induced residual stress, or simply, residual stress. Process-induced residual stress can be flow-induced or thermal-induced, with the latter being the dominant component.

In-cavity residual stress

While the part is still constrained in the mold cavity, the internal stress that accumulates during solidification is referred to as in-cavity residual stress. This in-cavity residual stress is the force that drives post-ejection part shrinkage and warpage.

Example

The shrinkage distribution described in [Warpage due to differential shrinkage](#) leads to a thermal-induced residual stress profile for an ejected part, as shown in the lower-left figure below. The stress profile in the upper-left figure is the in-cavity residual stress, in which the molded part remains constrained within the mold prior to ejection. Once the part is ejected and the constrained force from the mold is released, the part will shrink and warp to release the built-in residual stress (generally tensile stress, as shown) and reach an equilibrium state. The equilibrium state means that there is no external force exerting on the part and the tensile and compressive stresses over the part cross-section should balance with each other. The figures on the right side correspond to the case with a non-uniform cooling across the part thickness and, thereby, causing an asymmetric residual-stress distribution.

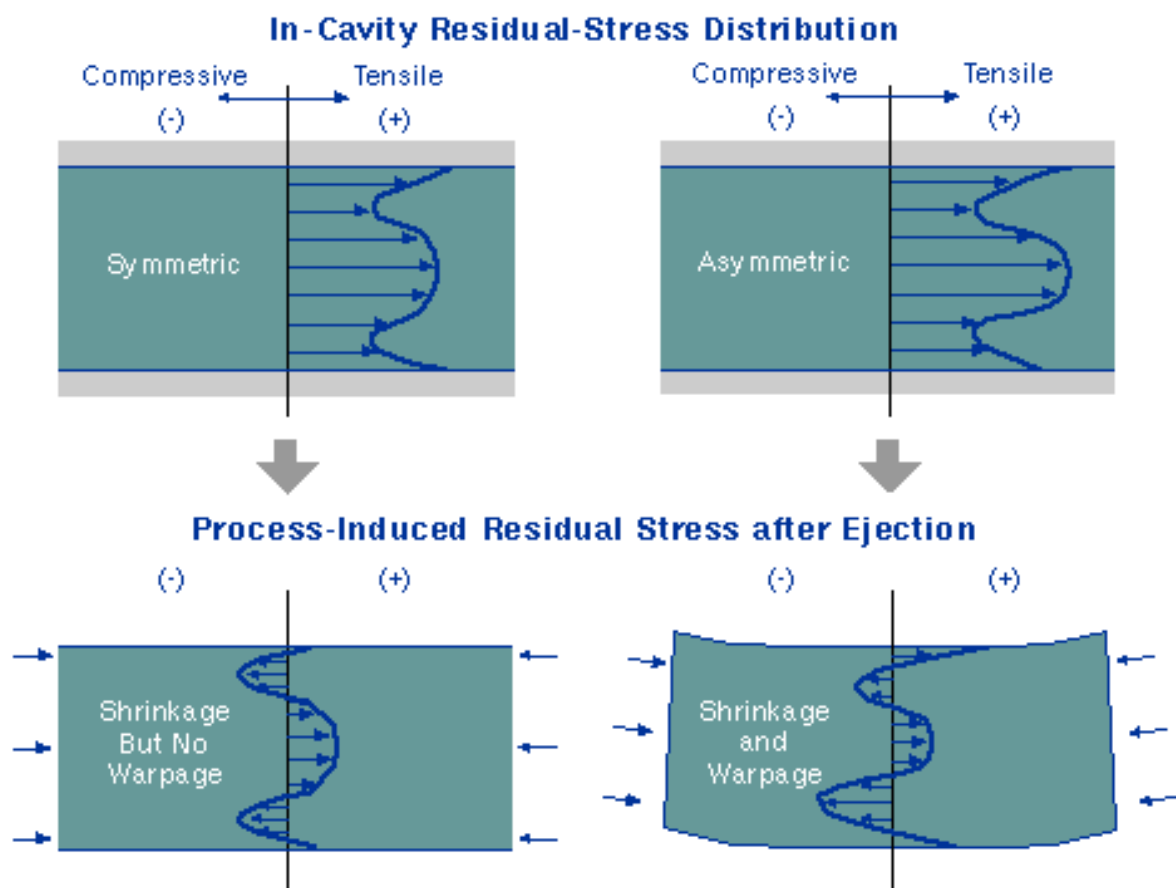


FIGURE 6. In-cavity residual stress profile (top) vs. process-induced residual stress profile and part shape after ejection (bottom).

Reducing thermal-induced residual stress

Conditions that lead to sufficient packing and more uniform mold-wall temperatures will reduce the thermal-induced residual stresses. These include:

- Proper packing pressure and duration
- Uniform cooling of all surfaces of the part
- Uniform wall-section thickness

Shrinkage and warpage

Why do they occur?

Shrinkage is inherent in the injection molding process. Shrinkage occurs because the density of polymer varies from the processing temperature to the ambient temperature (see [Specific volume \(pvT diagram\)](#)). During injection molding, the variation in shrinkage both globally and through the cross section of a part creates internal stresses. These so-called residual stresses (see [Residual stress](#)) act on a part with effects similar to externally applied stresses. If the residual stresses induced during molding are high enough to overcome the structural integrity of the part, the part will warp upon ejection from the mold or crack with external service load.

Shrinkage

The shrinkage of molded plastic parts can be as much as 20 percent by volume, when measured at the processing temperature and the ambient temperature. Crystalline and semi-crystalline materials are particularly prone to thermal shrinkage; amorphous materials tend to shrink less. When crystalline materials are cooled below their transition temperature, the molecules arrange themselves in a more orderly way, forming crystallites. On the other hand, the microstructure of amorphous materials does not change with the phase change. This difference leads to crystalline and semi-crystalline materials having a greater difference in specific volume (Δv) between their melt phase and solid (crystalline) phase. This is illustrated in [Figure 1](#) below. We'd like to point out that the cooling rate also affects the [fast-cooling pvT behavior of crystalline and semi-crystalline materials](#).

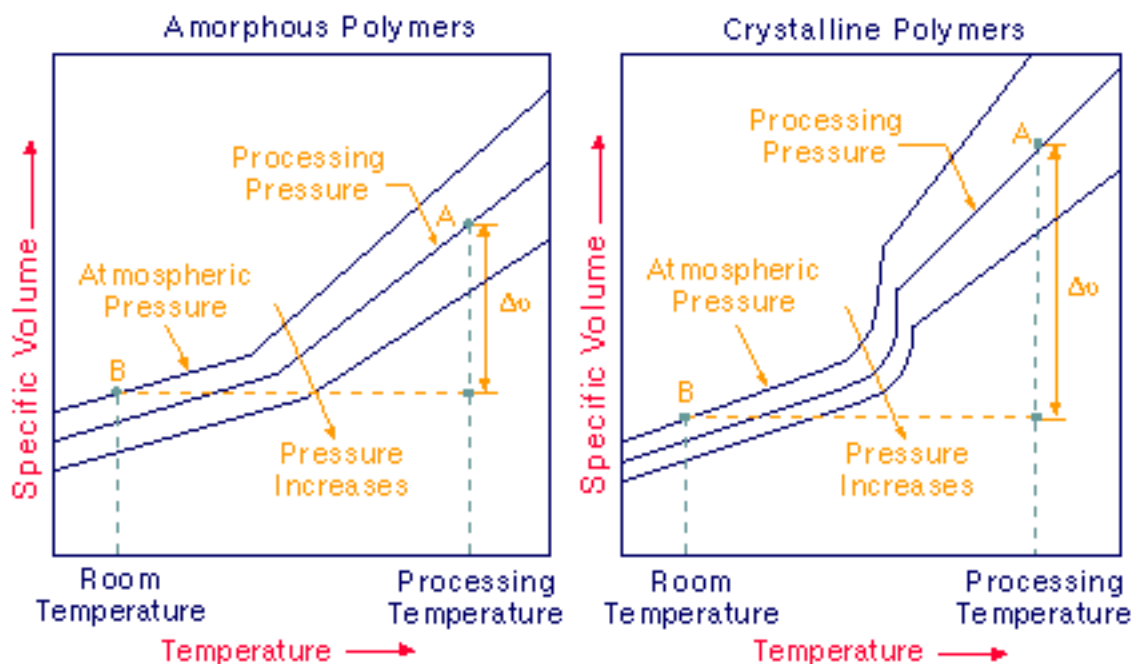


FIGURE 1. The pvT curves for amorphous and crystalline polymers and the specific volume variation (Δv) between the processing state (point A) and the state at room temperature and atmospheric pressure (point B). Note that the specific volume decreases as the pressure increases.

Causes of excessive part shrinkage

Excessive shrinkage, beyond the acceptable level, can be caused by the following factors. The relationship of shrinkage to several processing parameters and part thickness is schematically plotted in [Figure 2](#).

- low injection pressure
- short pack-hold time or cooling time
- high melt temperature
- high mold temperature
- low holding pressure.

Problems caused by part shrinkage

Uncompensated volumetric contraction leads to either [sink marks or voids](#) in the molding interior. Controlling part shrinkage is important in part, mold, and process designs, particularly in applications requiring tight tolerances. Shrinkage that leads to sink marks or voids can be reduced or eliminated by packing the cavity after filling. Also, the mold design should take shrinkage into account in order to conform to the part dimension. Part shrinkage predicted by C-MOLD offers a useful guideline for proper mold design.

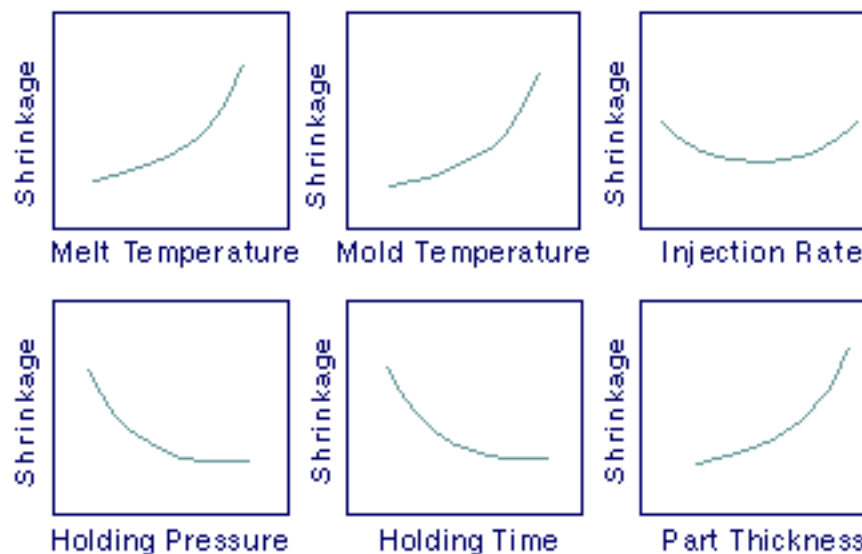


FIGURE 2. Processing and design parameters that affect part shrinkage

Warpage

Warpage is a distortion where the surfaces of the molded part do not follow the intended shape of the design. Part warpage results from molded-in [residual stresses](#), which, in turn, is caused by

differential shrinkage of material in the molded part. If the shrinkage throughout the part is uniform, the molding will not deform or warp, it simply becomes smaller. However, achieving low and uniform shrinkage is a complicated task due to the presence and interaction of many factors such as molecular and fiber orientations, mold cooling, part and mold designs, and process conditions.

Warpage due to differential shrinkage

Warpage in molded parts results from differential shrinkage. Variation in shrinkage can be caused by molecular and fiber orientation, temperature variations within the molded part, and by variable packing, such as over-packing at gates and under-packing at remote locations, or different pressure levels as material solidifies across the part thickness. These causes are described more fully below.

- Differences in filled and unfilled materials
- Non-uniform mold cooling across the part thickness or over the part
- Cooling rates that differ because of Part thickness variation
- Part geometry asymmetry or curvature

Differences in filled and unfilled materials

Differential shrinkage for filled and unfilled materials is shown in [Figure 3](#) below. When shrinkage is differential and anisotropic across the part and part thickness, the internal stresses created can lead to part warpage.

► Filled materials

For fiber-filled thermoplastics, reinforcing fibers inhibit shrinkage due to their smaller thermal contraction and higher modulus. Therefore, fiber-filled materials shrink less along the direction in which fibers align (typically the flow direction) compared to the shrinkage in the transverse direction. Similarly, particle-filled thermoplastics shrink much less than unfilled grades.

► Unfilled materials

On the other hand, if an unfilled molded part contains high levels of molecular orientation, shrinkage is anisotropic because aligned chains shrink to a greater extent in the direction of orientation.

► Liquid crystal polymers

For liquid crystal polymers (LCPs), the tightly ordered self-reinforcing structure tends to exhibit anisotropic shrinkage.

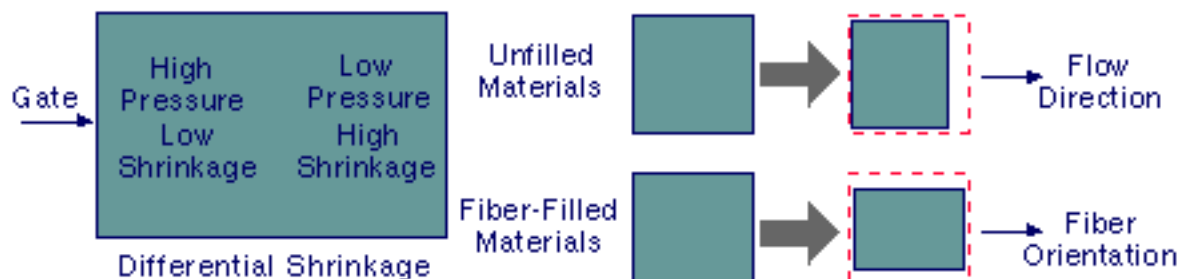
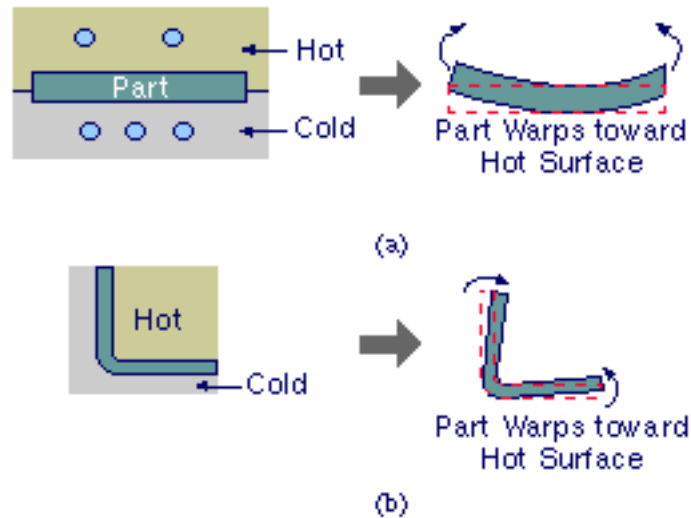


FIGURE 3. Differential shrinkage for both unfilled and filled materials

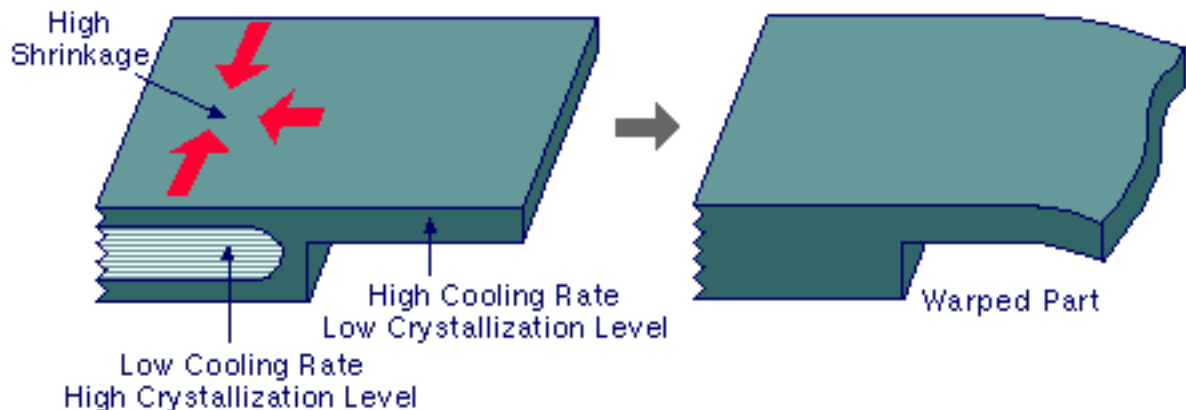
Non-uniform mold cooling across the part thickness

Non-uniform cooling in the part and asymmetric cooling across the part thickness from the mold cavity and core can also induce differential shrinkage. The material cools and shrinks inconsistently from the mold wall to the center, causing warpage after ejection.

**FIGURE 4.** Part warpage due to (a) non-uniform cooling in the part, and (b) asymmetric cooling across the part thickness.

Part thickness variation

Shrinkage increases as the wall thickness increases. Differential shrinkage due to non-uniform wall thickness is a major cause of part warpage in unreinforced thermoplastics. More specifically, different cooling rates and crystallization levels generally arise within parts with wall sections of varying thickness. This causes differential shrinkage, resulting in part warpage, as shown in [Figure 5](#) below.

**FIGURE 5.** Larger volumetric shrinkage due to the high crystallization level in the slow cooling areas (e.g., the thick sections) leads to differential shrinkage and thus part warpage

Part geometry asymmetry or curvature

Geometric asymmetry (e.g., a flat plate with a large number of ribs that are aligned in one direction or on one side of the part) will introduce non-uniform cooling and differential shrinkage that can lead to part warpage, as shown in [Figure 6](#) below.

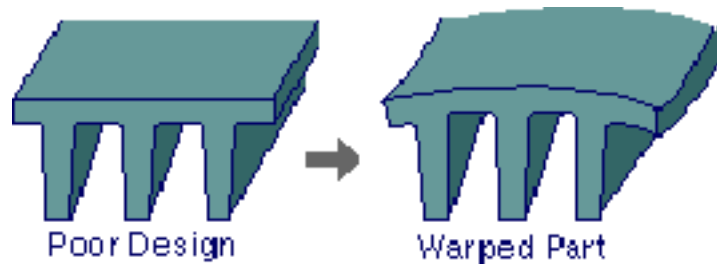


FIGURE 6. The poor cooling of the mold wall on the ribbed side causes a slower cooling of the material on that one side, which can lead to part warpage

Design rules for shrinkage and warpage

Guidelines

You can reduce or control shrinkage and warpage by properly designing the part, mold, and process, as well as through careful material selection. The following design rules provide some guidelines for developing low-shrinkage, warp-free parts.

Wall thickness

Avoid non-uniform wall thickness or design a transition length of three times the thickness of the thinner region, as illustrated below.

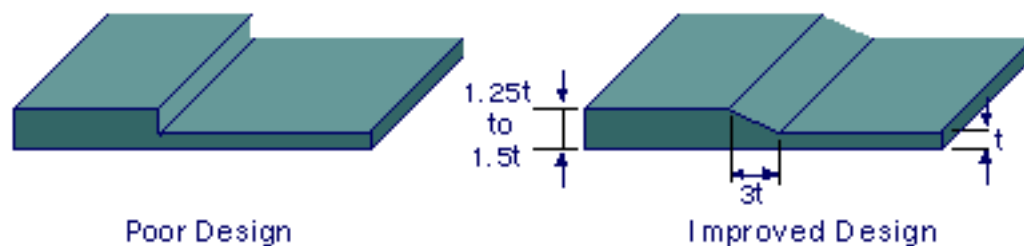


FIGURE 1. Part design with thickness variation.

► Thick sections

Alter the design to replace thick sections that cause significant shrinkage and lead to sink marks or internal voids. A thin, uniform wall with ribs (shown in [Figure 2](#) below) provides for uniform shrinkage, strength to weight ratio, and cost effectiveness.

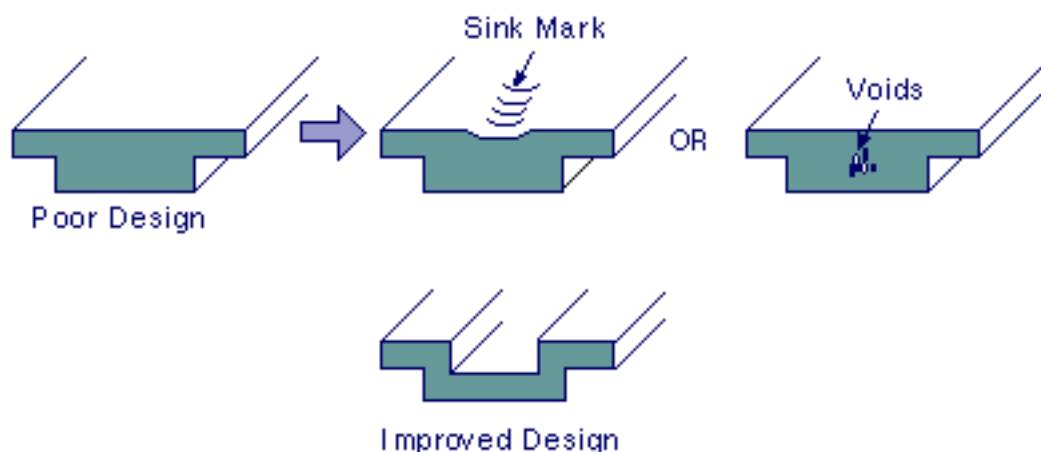


FIGURE 2. In most applications, a thin uniform wall with ribs is preferred to a thick wall.

Balanced filling

Design the delivery system to provide a balanced filling pattern with a constant melt-front velocity.

Packing pressure

While high packing pressure can help reduce the shrinkage, it also potentially increases the level of residual stress and the clamp force requirement. For a better process design, use a proper packing pressure level, allow sufficient packing time, and remove the pressure after the gate freezes off. The packing pressure should be able to deliver additional material to compensate for the volumetric shrinkage within the part.

Cooling system

Design the cooling system to apply uniform, balanced cooling, both across the thickness and throughout the part.

Residual stress

Reduce residual stress and molecular or fiber orientation by increasing the melt temperature, mold-wall temperature, fill time, and cavity thickness, or by decreasing the packing pressure and flow path.

Design Example: Reducing shrinkage and warpage

Purpose

This example shows how to reduce shrinkage and warpage for a molded plastic part. It illustrates the role of computer simulation at each step in the process.

Software used

The software used for this example is C-MOLD Performance Solution, an integrated three-dimensional analysis that simulates the fluid flow and heat transfer of compressible thermoplastic melt as it fills the cavity, cools, and solidifies. This software package consists of C-MOLD Filling, Post-Filling, Cooling, Residual Stress and Fiber Orientation, and Structural Analyses. C-MOLD Performance Solution predicts the dimensional stability of the part, such as shrinkage and warpage. It helps engineers identify mechanisms, such as unbalanced cooling or non-uniform shrinkage, that cause warpage. Based on the simulation results, engineers can improve dimensional conformity and achieve tolerance requirements.

Description

The example used in this document goes several steps beyond other examples under Design > Using CAE, which demonstrate strategic design development for the early phases of the plastic molding process in improving the filling pattern, reducing the injection pressure, and optimizing the gate location. This example uses C-MOLD Performance Solution, for more comprehensive design development, encompassing the latter phases of the molding process to reduce the part shrinkage and warpage. As detailed in [residual stress](#), part [shrinkage and warpage](#) is a complex problem that involves dynamic interactions of the following factors:

1. Histories of temperature, pressure, crystallization, etc., of the material elements.
2. Non-uniform shrinkage in planar and gap-wise directions within the part.
3. Different material properties (anisotropy) due to molecular and fiber orientations.
4. Effects of part stiffness as well as mold constraint and deformation.

NOTE: Click on an image below to open a full-size version.

► Given

Part and tool designs for a hand-held bar-code scanner. The bottom half of the scanner housing is used in this example.



► Objective

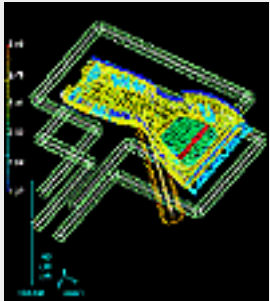
To determine dimensional conformity and solutions to reduce shrinkage and warpage.

► Design rules

- Design for uniform and efficient mold cooling
- Apply equal packing to avoid differential shrinkage that causes warpage.

DESIGN1: ORIGINAL DESIGN

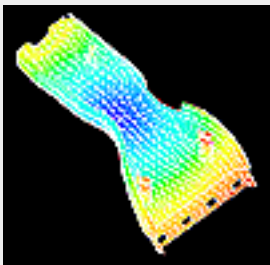
Description: Center gate



[Enlarge](#)

► Details

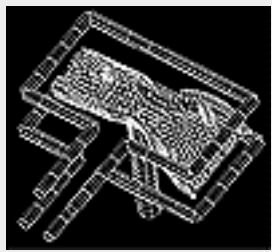
The part has a center gate with a hot-runner drop upstream from the single gate. The thickness varies from 2.54 mm across the center portion to 3.2 mm on the larger radii that run the length of each side. The periphery of the open edge is designed to be 1.27 mm thick.



[Enlarge](#)

► Process conditions

- Fill time: 2 seconds
- Variable ram-speed profile
- Melt temperature: 313 °C
- Coolant temperature: 85 °C
- Pack/hold pressure: 75 MPa
- Timer for hold pressure: 2.5 seconds
- Post-fill time: 23 seconds
- Mold open time: 7 seconds


[Enlarge](#)

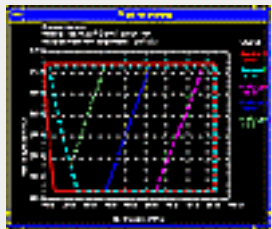
► Hints, Tips, and Techniques:

- Mesh density: 30 to 50 elements along the largest dimension of the part
- Finer mesh for critical regions (e.g., gate and living hinge)
- Variable mesh density for fine-to-coarse mesh transition
- Maintain 2:1 (length / diameter) aspect ratio for cooling channel elements


[Enlarge](#)

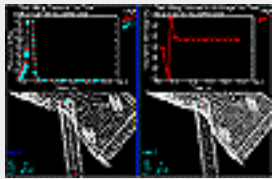
► Action

1. Run Process Estimator or Dr. C-MOLD to quickly identify the process window.
2. Launch a full-scale analysis.


[Enlarge](#)

► Analysis results

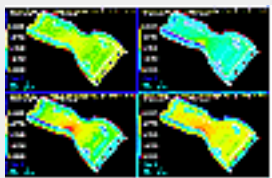
Given the short timer for hold pressure (2.5 seconds) as shown in the X-Y plot of pressure versus time at the gate (left figure), C-MOLD Filling/Post-Filling analysis predicts an increase of volumetric shrinkage at the gate node after the removal of packing pressure (right figure). This indicates that the pack/hold time might not be sufficient and a portion of the gate remains molten.


[Enlarge](#)

► Analysis results

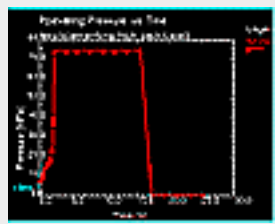
The results clearly exhibit the sudden increase of volumetric shrinkage and its propagation around the part center due to premature pressure removal:

- at the end of fill (upper left)
- shortly before removal of packing pressure (upper right)
- shortly after removal of packing pressure (lower left)
- at the end of post-fill (lower right)


[Enlarge](#)

► Remedy

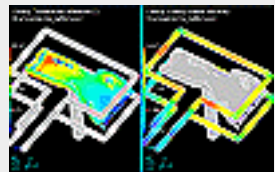
Increase the timer for hold pressure so that it corresponds to the gate freeze-off time.



[Enlarge](#)

► Hints, Tips, and Techniques

- Avoid early removal of pack pressure
- Back flow leads to higher shrinkage. Detect back flow by:
 - Averaged velocity or flow rate in 1-D element
 - Sudden jump in volumetric shrinkage at gate region
 - Negative pressure gradient from entrance to gate
 - Temperature profile or frozen layer fraction of gate element
- Avoid unnecessary packing after gate freeze-off



[Enlarge](#)

► Analysis results

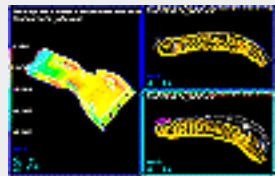
Unbalanced cooling: C-MOLD Cooling results show a large temperature difference (shown in red in the left-hand figure) around the corners of the part.

High coolant temperature (85 °C) leads to negative cooling channel efficiency around the coolant entrances and exits (right-hand figure).

Why? Heat is being transferred from the cooling channels to the ambient, which is at the room temperature (25 °C).

► Remedy

Add more cooling devices (like baffles) to promote a more uniform and efficient cooling.



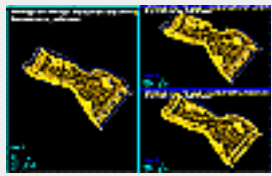
[Enlarge](#)

► Hints, Tips, and Techniques

Take caution to interpret C-MOLD shrinkage and warpage in terms of displacement.

NOTE: Part shape and distance between points are unique solutions, regardless of how the part is anchored in space.

- Displacement and relative displacement are not unique and are dependent on how the part is anchored. For example, the Y-displacement based on the "Best-Fit Transformation" (upper right) is different from the one with "User-Defined Anchor Plane" (lower right).
- Displacement from C-MOLD is based on best-fit transformation by default.
- Best-fit transformation is automatically provided for easier visual comparison of part deformation from the desired shape.
- The "User-Defined Anchor Plane" is for the actual measurement comparison.
- Different anchor planes yield different values of displacement


[Enlarge](#)

► Analysis results

The C-MOLD analysis predicts a large shrinkage and warpage contributed by both non-uniform shrinkage and unbalanced cooling. Since the upper half has a similar warpage in the opposite direction (not shown), the total gap at this scanning end will exceed the specified limit.

Problem: This is unacceptable because it could cause electromagnetic interface (EMI) leakage.


[Enlarge](#)

► Hints, Tips, and Techniques

Uniformity is the key for reducing warpage

- Thickness
- Cooling
- Packing
- Properties
- Shrinkage

Pack/hold pressure and duration affect shrinkage

- Packing reduces shrinkage

But high packing pressure causes:

- Problem in de-molding (part sticks to the mold)
- High clamp force requirement
- Possible density variation within the part
- Mold deformation

► Remedy

In addition to increasing the packing time, add more baffles to promote a more uniform and efficient cooling.

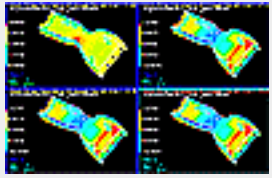
DESIGN 2: MODIFIED DESIGN

Design change: Added baffles, increased packing time


[Enlarge](#)

► Analysis results

By adding three baffles to the cooling system, the predicted temperature difference across the part thickness is drastically reduced and becomes more uniform. The majority of the part has a temperature difference smaller than 10 °C.

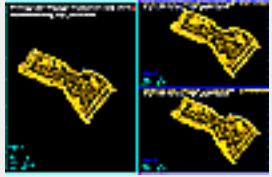


[Enlarge](#)

► Analysis results

With a longer packing time (15 seconds), the predicted volumetric shrinkage not only reduces in magnitude but also becomes more uniform. The results show the volumetric shrinkage predictions

- at the end of fill (upper left)
- shortly before removal of packing pressure (upper right)
- shortly after removal of packing pressure (lower left)
- at the end of post-fill (lower right).



[Enlarge](#)

► Analysis results

The predicted part shrinkage and warpage due to non-uniform shrinkage and unbalanced cooling have been reduced significantly. The dimensional conformance is met with the modified design.

Air traps

What is an air trap?

An air trap is air that is caught inside the mold cavity. It becomes trapped by converging polymer melt fronts or because it failed to escape from 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 race-tracking (the tendency of polymer melt to flow preferentially in thicker sections), caused by a large thickness ratio.

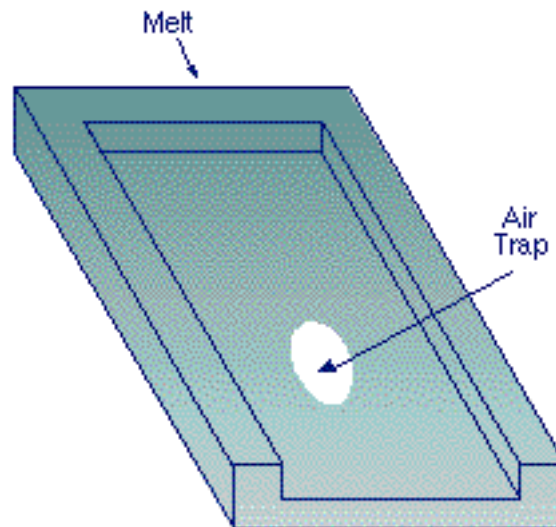


FIGURE 1. Air trap locations indicated by the computer-predicted melt-front advancements. (To replay the animation, click [Replay](#))

Problems caused by air traps

Entrapped 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. To eliminate air traps, you can modify the filling pattern by reducing the injection speed, enlarging venting, or placing proper venting in the cavity.

In the figure below, an air trap results from race-tracking in a part with a large thickness ratio. In this case, the entrapped air can be vented by changing the thickness ratio or by placing a vent (e.g., via an ejector pin) at the air trap location.

Remedies

Alter the part design



Reduce the thickness ratio.

This will minimize the race-tracking effect of polymer melt.

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 flash at the edge of the molding. The recommended vent size is 0.025 mm (0.001 inch) for crystalline polymers, and 0.038 mm (0.0015 inch) for amorphous polymers.

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.

Black specks/black streaks

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.

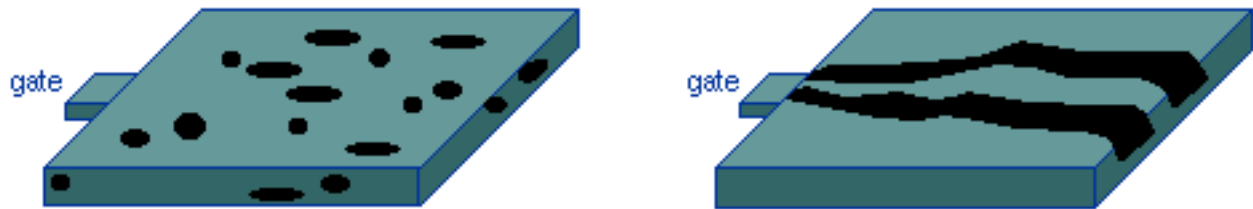


FIGURE 1. Black specks (left) and black streaks (right)

Causes of black specks/black streaks

Black specks and black streaks are caused by overheated (degraded, burned) material or by contamination of the resin.

► 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.

► 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.

Other defects resulting from the same causes

- [Brittleness](#)
- [Burn marks](#)
- [Discoloration](#)

Remedies

Handle the material carefully

- ➡ **Make sure 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.**

Alter the mold design

- ➡ **Clean the ejectors and slides. The streaks could be 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.**

Select a proper machine

- ➡ **Size a proper injection machine for a specific mold.**
The typical shot size should be between 20 and 80 percent of machine injection capacity. For temperature-sensitive materials, the range should be narrowed down 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.**

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'll be used for parts that are in black or for which such defects are acceptable.

Brittleness

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 1. Degraded part tends to be brittle and break easily

Causes of brittleness

Brittleness is caused by material degradation due to

- Improper drying conditions
- Improper temperature setting
- Improper runner system and gate design
- Improper screw design
- Weld line weakness

Other defects resulting from the same causes

- Black specks/black streaks
- Burn marks
- Discoloration

Remedies

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.

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.

Select a proper machine/machine component

- ➡ **Get a better screw design for the material you're using 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.

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.** since shear heating can result in material degradation.
- ➡ **Within the limit not to overheat the material, increase melt temperature, mold temperature or injection pressure if the weld line has a tendency to crack.** See [Weld lines and meld lines](#) for more information.

Burn marks

What is a burn mark?

Burn marks are small, dark or black spots that appear near the end of the flow path of a molded part or in the blind area where the air trap forms.



FIGURE 1. Burn marks

Causes of burn marks

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, and which will cause the polymer to degrade on the surface near the end of the flow path or the blind area.

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.

Other defects resulting from the same causes

- [Black specks/black streaks](#)
- [Brittleness](#)
- [Discoloration](#)

Remedies

Alter the mold design



Place an adequate venting system throughout the mold to help vent out the entrapped air.

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 inches) for crystalline polymers, and 0.038 mm (0.0015 inches) 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.

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.

Delamination

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.

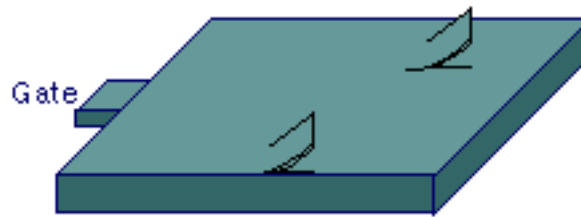


FIGURE 1. Delamination causes layer-wise peel-off on the surface of a molded part

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

Remedies

Change the material preparation

- ➡ **Avoid using foreign material or contaminated regrind material in the molding process.**

Alter the mold design

- ➡ **Smooth all of the corners at the gate and runner.**
Sharp corners can tear apart melt flow and cause lamination.

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 over-using 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 lamination 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.

Dimensional variation

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 1. Dimensional variation is an unexpected change of part dimension

Causes of dimensional variation

Dimensional variation can be caused by

- Unstable machine control
- A narrow process window
- Improper process conditions settings
- A broken check ring (within the injection unit)
- Unstable material property

Remedies

Improve the material preparation



- ➡ **Contact the material vendor and change the material lot if the material has patch-to-patch 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.

Change a mold design/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 dimension to assure a smooth melt flow into the cavity.

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.**

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.**

Discoloration

What is discoloration?

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

Causes of discoloration

This defect can be caused by either material degradation or contamination from the following problems:

- The material staying in the barrel too long.
- The barrel temperature being too high, causing the color to change.
- Contamination caused by reground material, different color material, or foreign material.

Other defects resulting from the same causes

- [Black specks/black streaks](#)
- [Brittleness](#)
- [Burn marks](#)

Remedies

Handle the material carefully



Maintain proper housekeeping for origin materials and regrind materials storage to avoid contaminated materials.

Alter the mold design



Add an adequate venting system.

To avoid discoloration (or [burn mark](#)) due to poor venting or [air traps](#), use the recommended venting size: 0.025 mm (0.001 inches) for crystalline polymers, and 0.038 mm (0.0015 inches) for amorphous polymers.

Select a proper machine



Use a different size injection molding machine.

The typical shot size should be between 20 and 80 percent of machine injection capacity. For temperature-sensitive materials, the range should be narrowed down, depending on the material. 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.

Adjust the molding conditions



Clean the hopper completely.

It's important to avoid foreign material or different color materials mixing together before molding.



Purge the injection unit completely if there is any material changing.



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.

Fish eyes

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.

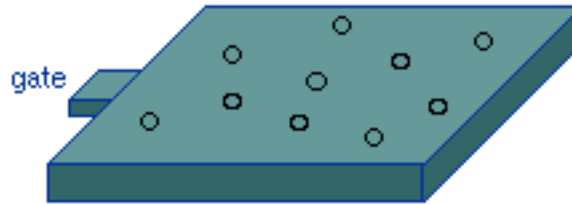


FIGURE 1. Unmelted materials in the melt stream causes fish eyes

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, and 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 setting are set too low, there might not be enough frictional heating to melt the material completely in the barrel before the injection.

Remedies

Improve the material preparation



Limit or eliminate regrind for practical molding, depending on part quality requirements.

Adding ten percent 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.

Adjust the molding conditions

Material suppliers usually provide the information about barrel temperature, back pressure, and screw rotation speed for specific materials. If you've 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.

Flash

What is flash?

Flash is a defect where excessive material is found at locations where the mold separates, notably the parting surface, movable core, vents, or venting ejector pins.



FIGURE 1. Flash

Causes of flash

► Low clamp force

If the clamp force of the injection machine is too weak to hold the mold plates together during the molding process, flash will occur.

► 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.

► Molding conditions

Improper molding conditions, such as a high melt temperature (which makes a thinner melt) or high injection pressure, will cause flash.

► Improper venting

An improperly designed venting system, a very poor venting system, or a venting system that is too deep, will cause flash.

Remedies

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 inches) for crystalline polymers, and 0.038 mm (0.0015 inches) 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.

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.

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 slow down the injection speed.

Flow marks

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 1. Flow marks

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 can usually 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 due to cooling of the flow front portion on a cavity wall and the repeated phenomena of "getting over" and cooling with the subsequent melt. This is discussed in [Ripples](#).

Remedies

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.**

Adjust the molding conditions

 **Increase the injection pressure and packing pressure.**

 **Increase the barrel and nozzle temperature.**

 **Increase the mold temperature.**

Hesitation

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.

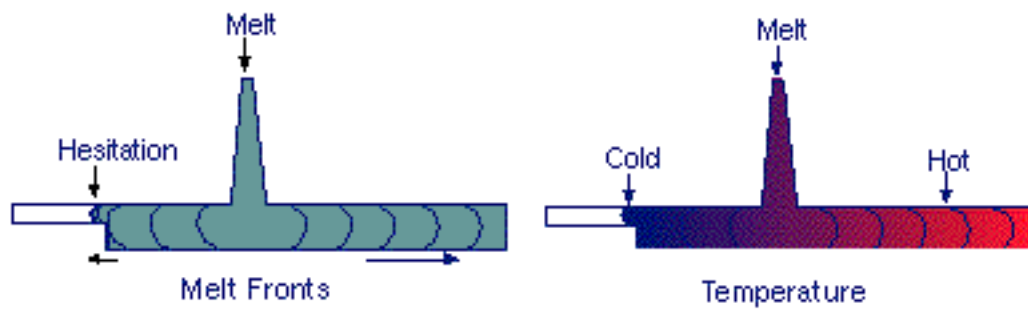


FIGURE 1. Hesitation results from stagnation of polymer melt flow

Problems caused by hesitation

When polymer melt is injected into a cavity of variable thickness, it tends to fill the thick, 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 1](#)). 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.

Remedies

When troubleshooting the cause of hesitation in your part, you'll need to rethink both your part and mold design. You can also try fine-tuning your processing conditions.

Alter the part design



Reduce part thickness variation.

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 2](#) shows that in a poor design, hesitation results from stagnation of polymer melt flow; moving the gate away from the thin section reduces hesitation.

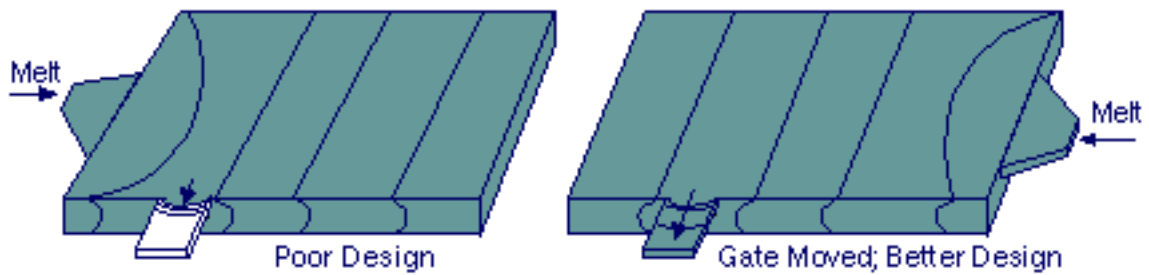


FIGURE 2. Hesitation resulting from improper gate position

Adjust the molding conditions

➡ **Increase the melt temperature and/or the injection pressure.**

Jetting

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, snake-like jetting stream causes contact points to form between the folds of melt in the jet, creating small-scale "welds" (see [Figure 1](#)).

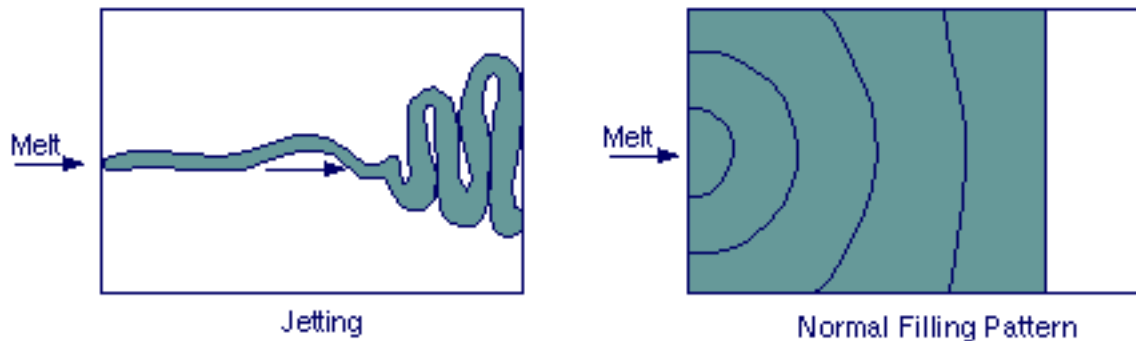


FIGURE 1. Jetting vs. normal filling pattern.

Effects of jetting

Jetting leads to part weakness, surface blemishes, and a multiplicity 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 above.

Remedies

Alter the mold design

You'll often find that the trouble lies with the gate design.



Direct the melt against a metal surface.



Use an overlap gate or a submarine gate as shown in [Figure 2](#) below.

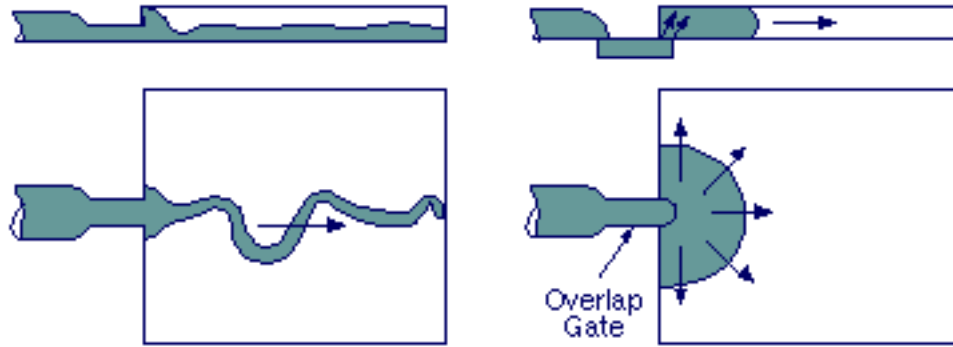


FIGURE 2. Using an overlap gate to avoid jetting

➡ **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. This reduces the melt shear stress and shear rate.

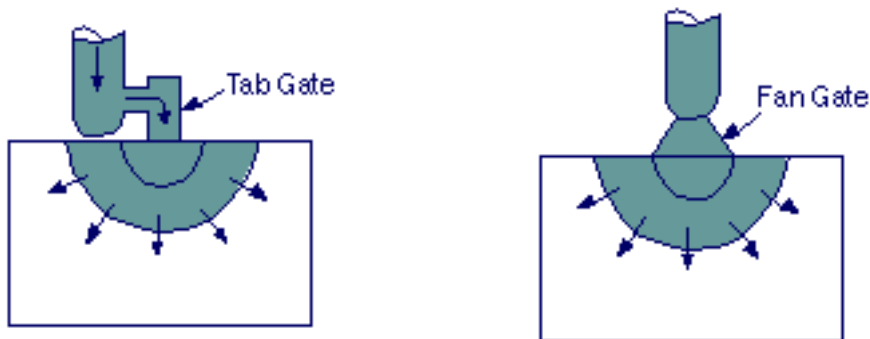


FIGURE 3. Using a tab gate (left) and a fan gate (right) to avoid jetting

➡ **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.

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 4](#) below illustrates this technique.

➡ **Adjust the barrel temperature to increase or decrease the melt temperature incrementally.**

The explanation for this is not well understood, but might be related to the die-swell effect and the change of the melt properties (such as viscosity and surface tension). For example, for most polymers, die swell increases as temperature decreases, while some materials, such as rigid PVC, exhibit increasing die swell as temperature increases.

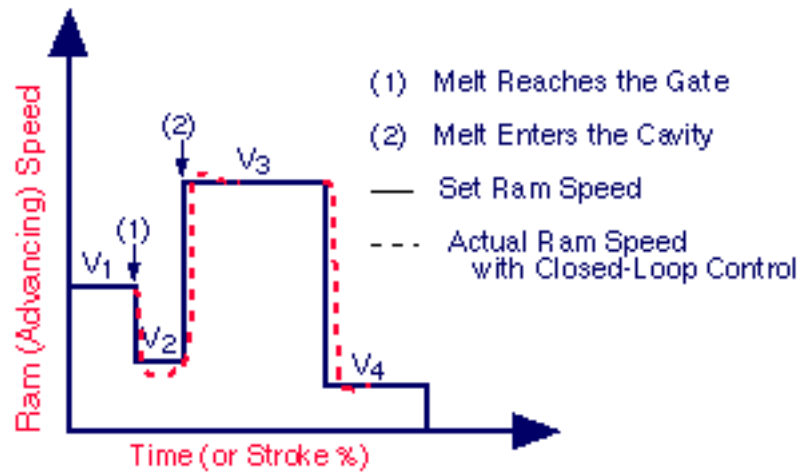


FIGURE 4. Adjust ram-speed profile to avoid jetting.

Ripples

What are ripples?

Ripples are the wavelets or small fingerprint-like waves near the edge or at the end of the flow.



FIGURE 1. Ripples

Cause of ripples

According to a recent visual analysis using a glass-inserted mold, the ripple defect is due to 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 the 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.

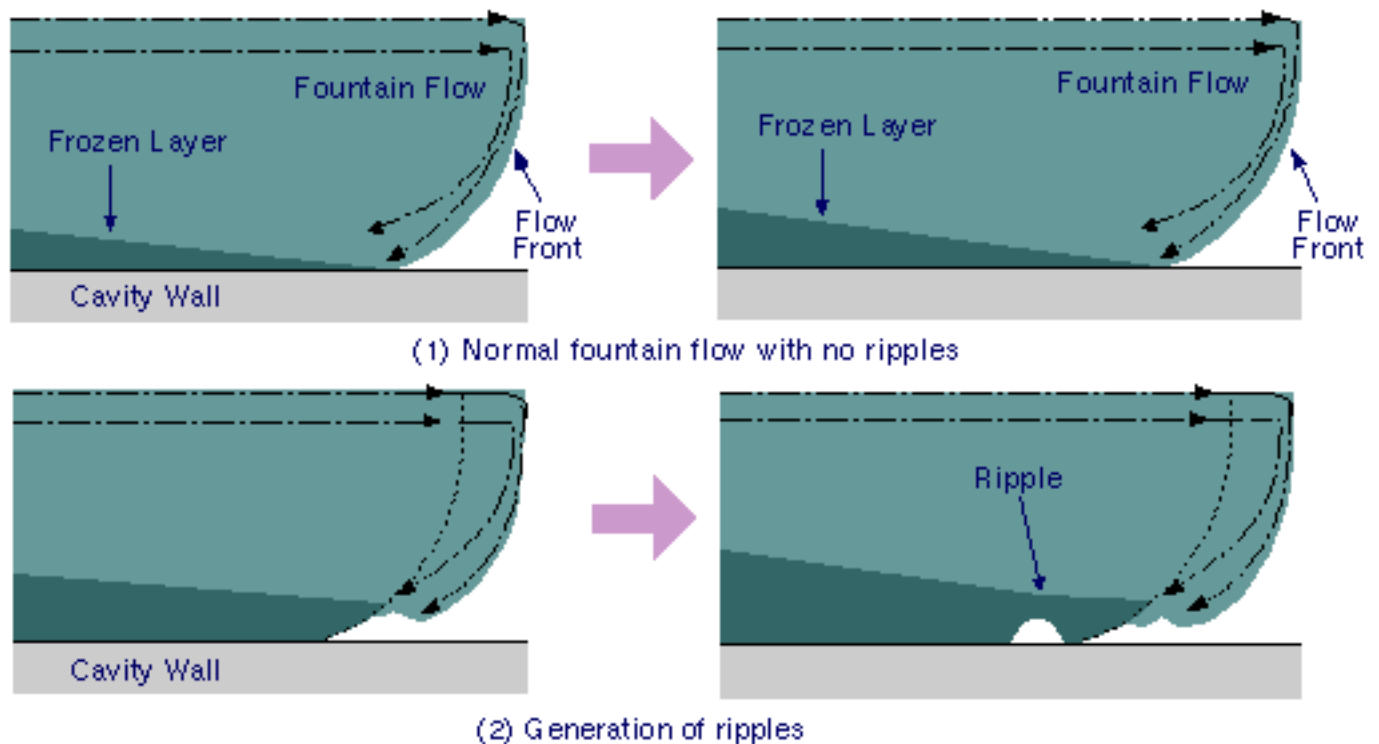


FIGURE 2. (1) Normal filling with no ripples. (2) Generation of ripples with cold mold wall and low melt-front velocity.

Remedies

Actions that increase the melt-front velocity or mold/melt temperature will help to eliminate the ripples.

Modify the part design

- ➡ **Increase the part thickness.**

Change the mold design

- ➡ **Make sure the runner system, including the sprue, runners, and gates, is adequate for the specific part.**
- ➡ **Place an adequate 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 inches) for crystalline polymers, and 0.038 mm (0.0015 inches) for amorphous polymers.

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 machine's capacity. The operating injection pressure should normally be limited to 70 to 85 percent 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 due to prolonged exposure at an elevated temperature.

Short shot

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)-most commonly at thin sections or extremities.

Causes of short shot

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 non-return (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.

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.

Alter the part design

It's 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).**

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.**

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 percent 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're 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.

Silver streaks

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.

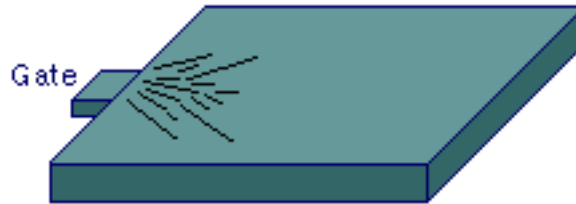


FIGURE 1. Silver streaks

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 couple of 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

Improper barrel temperature setting may degrade polymer molecules, and they will begin to char.

► Shot volume

If the shot size is below 20 percent 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.

Remedies

Handle the material carefully

- ➡ **Dry the material properly before molding, according to the resin supplier's instructions.**

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 inches) for crystalline polymers, and 0.038 mm (0.0015 inches) for amorphous polymers.

Adjust the molding conditions

These 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 percent 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's important to allow air and steam to escape easily.
- ➡ **Decrease the melt temperature, injection pressure, or injection speed.**

Sink marks and voids

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.

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 in extrusion 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 1](#) illustrates this phenomenon.

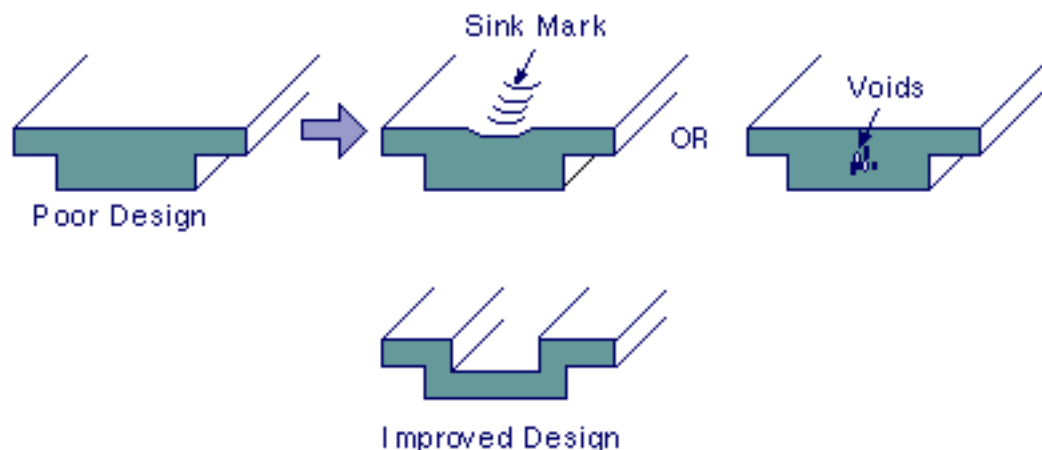


FIGURE 1. Sink marks and voids are created by material shrinkage without sufficient compensation.

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.

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 2 illustrates this technique.

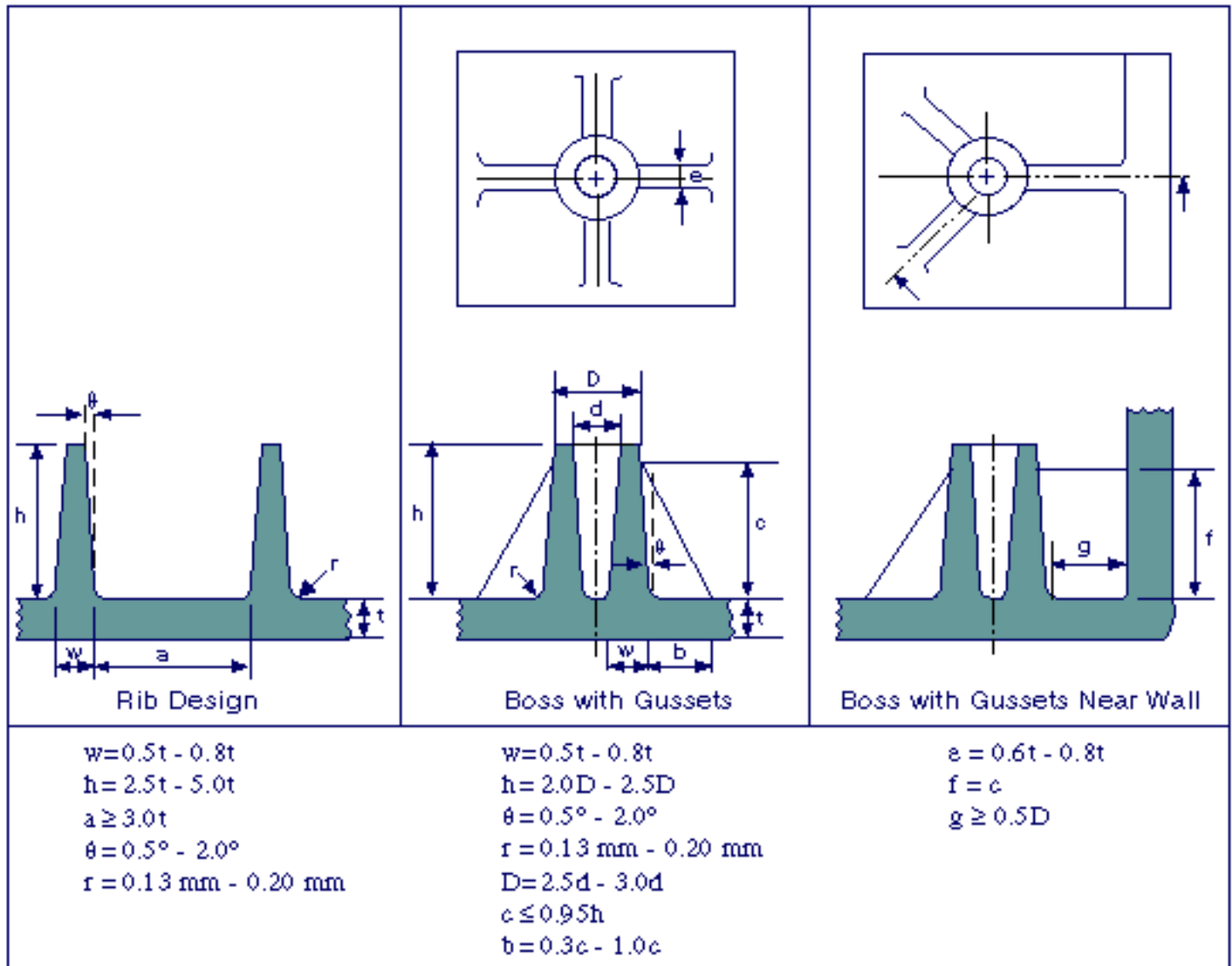


FIGURE 2. Sink marks can be eliminated by creating a design, rib, serrations.

- ➡ 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 percent of the attached (base) wall thickness.

Figure 3 shows the dimensions we prescribe.

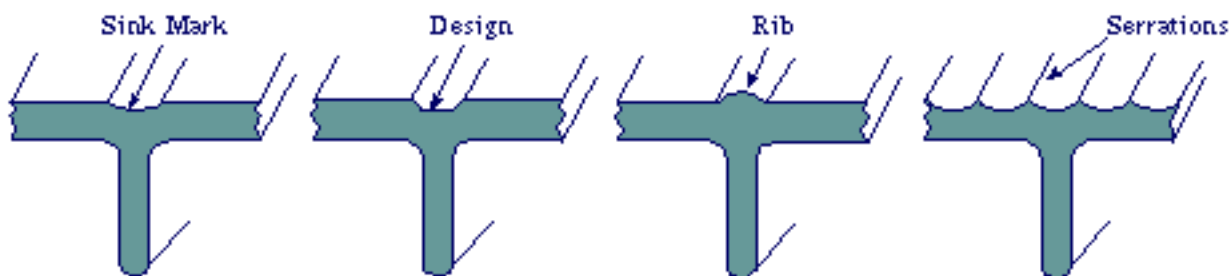


FIGURE 3. Recommended dimensions for ribs, bosses, and gussets

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 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.

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 inches).
- ➡ **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.**

Weld lines and meld lines

What are weld 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 can't be avoided, position them at low-stress and low-visibility areas by adjusting the gate position and dimension. Improve the strength of weld and meld lines by increasing the local temperature and pressure at their locations.

How to tell the difference between weld and meld lines

Traditionally, the "meeting angle" is used to differentiate weld lines and meld lines. As illustrated in [Figure 1](#) below, a meeting angle, θ , smaller than 135° produces a weld line; greater than 135° , a meld line. Note that the weld line surface mark disappears when the meeting angle reaches 120° to 150° . 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.

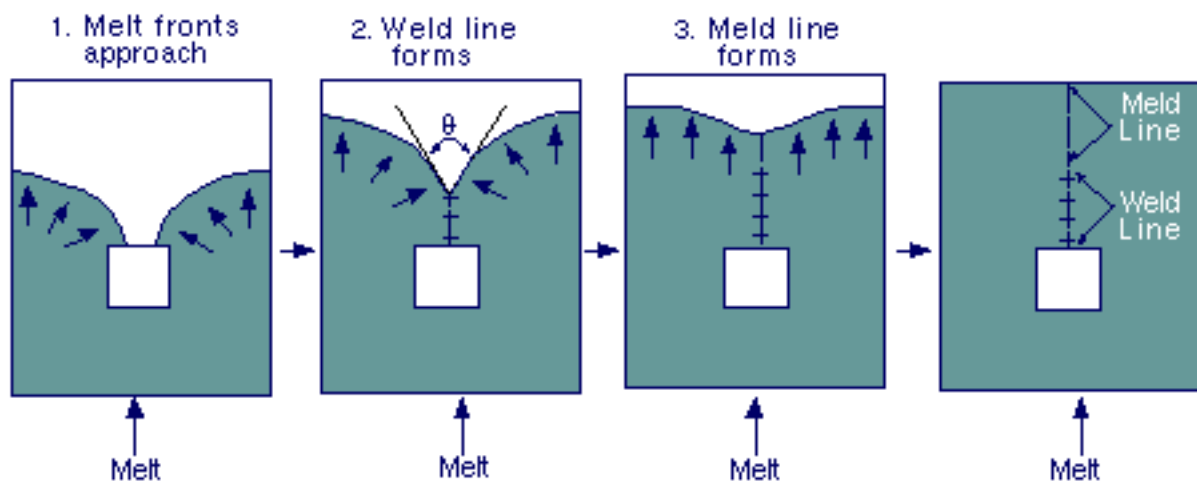


FIGURE 1. Weld and meld lines

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 illustratedn [Figure 2](#) below.

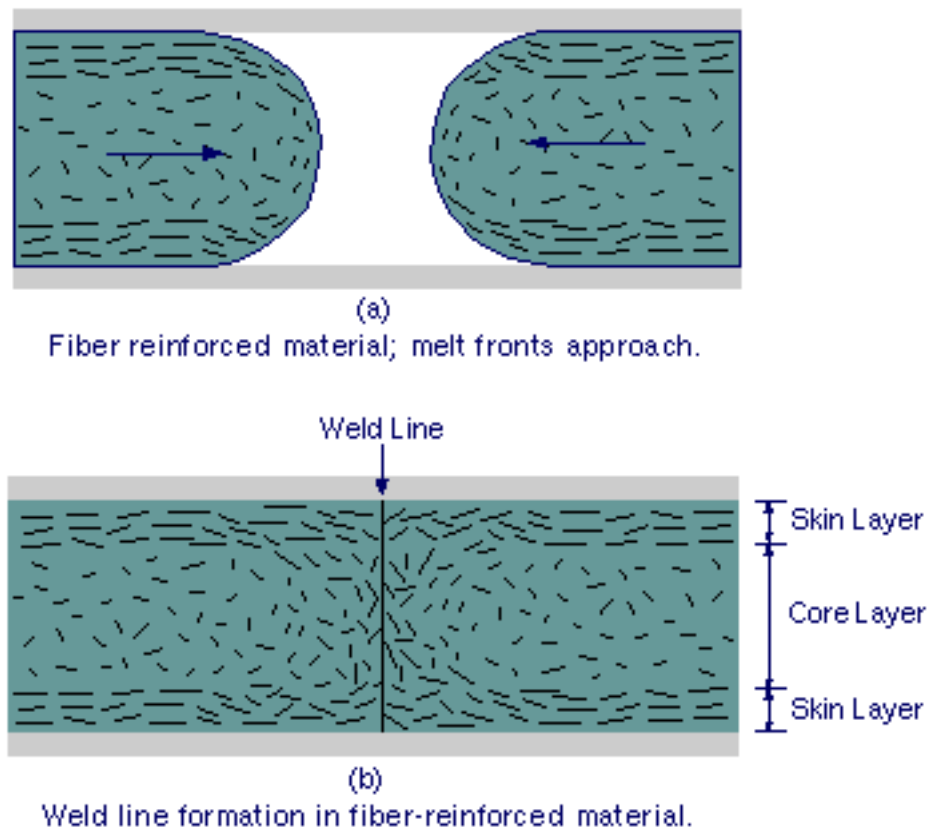


FIGURE 2. Fiber distribution parallel to the weld line leads to a weaker bond

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 percent 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 than 10°C between the two emerging melt 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 3. Changing the weld-line position by modifying the delivery system.

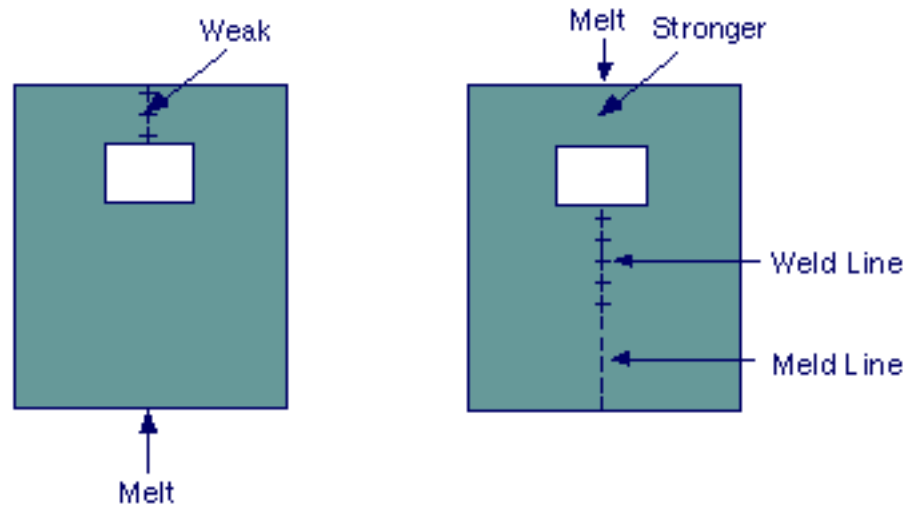


FIGURE 4. Improving the weld-line position by modifying the delivery system.

Remedies

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 and dimension or decrease the part thickness ratio.**

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 high temperature and under high packing pressure.**

Adjust the molding conditions

- ➡ **Increase the melt temperature, injection speed, or injection pressure.**

Injection Molding Overview

Process

Injection molding is a cyclic process of forming plastic into a desired shape by forcing the material under pressure into a cavity. The shaping is achieved by cooling (thermoplastics) or by a chemical reaction (thermosets). It is one of the most common and versatile operations for mass production of complex plastics parts with excellent dimensional tolerance. It requires minimal or no finishing or assembly operations. In addition to thermoplastics and thermosets, the process is being extended to such materials as fibers, ceramics, and powdered metals, with polymers as binders.

Applications

Approximately 32 percent by weight of all plastics processed go through injection molding machines. Historically, the major milestones of injection molding include the invention of the reciprocating screw machine and various new alternative processes, and the application of computer simulation to the design and manufacture of plastics parts.

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.

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 due to the slow heating rate through pure conduction only. The reciprocating screw can plasticize the material more quickly and uniformly with its rotating motion, as shown in [Figure 1](#). In addition, it is able to inject the molten polymer in a forward direction, as a plunger.

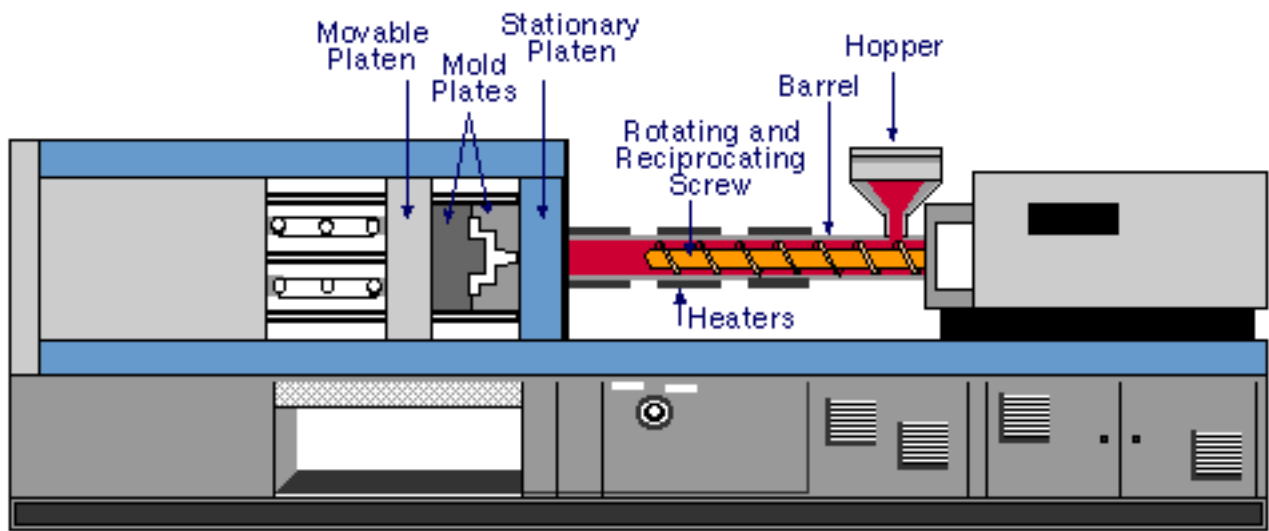


FIGURE 1. The reciprocating-screw injection machine

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.

Alternative injection molding processes

During the past two 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
- 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

Computer simulation of injection molding processes

Because of these extensions and their promising future, computer simulation of the process has also expanded beyond the early "lay-flat," empirical cavity-filling estimates. Now, complex programs simulate post-filling behavior, reaction kinetics, and the use of two materials with different properties, or two distinct phases, during the process.

The [Simulation](#) section provides information on using C-MOLD products. Among the [Design](#) topics are several examples that illustrate how you can use CAE tools to improve your part and mold design and optimize processing conditions.

Co-injection (sandwich) molding

Overview

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 1 shows the steps involved in the co-injection molding process.

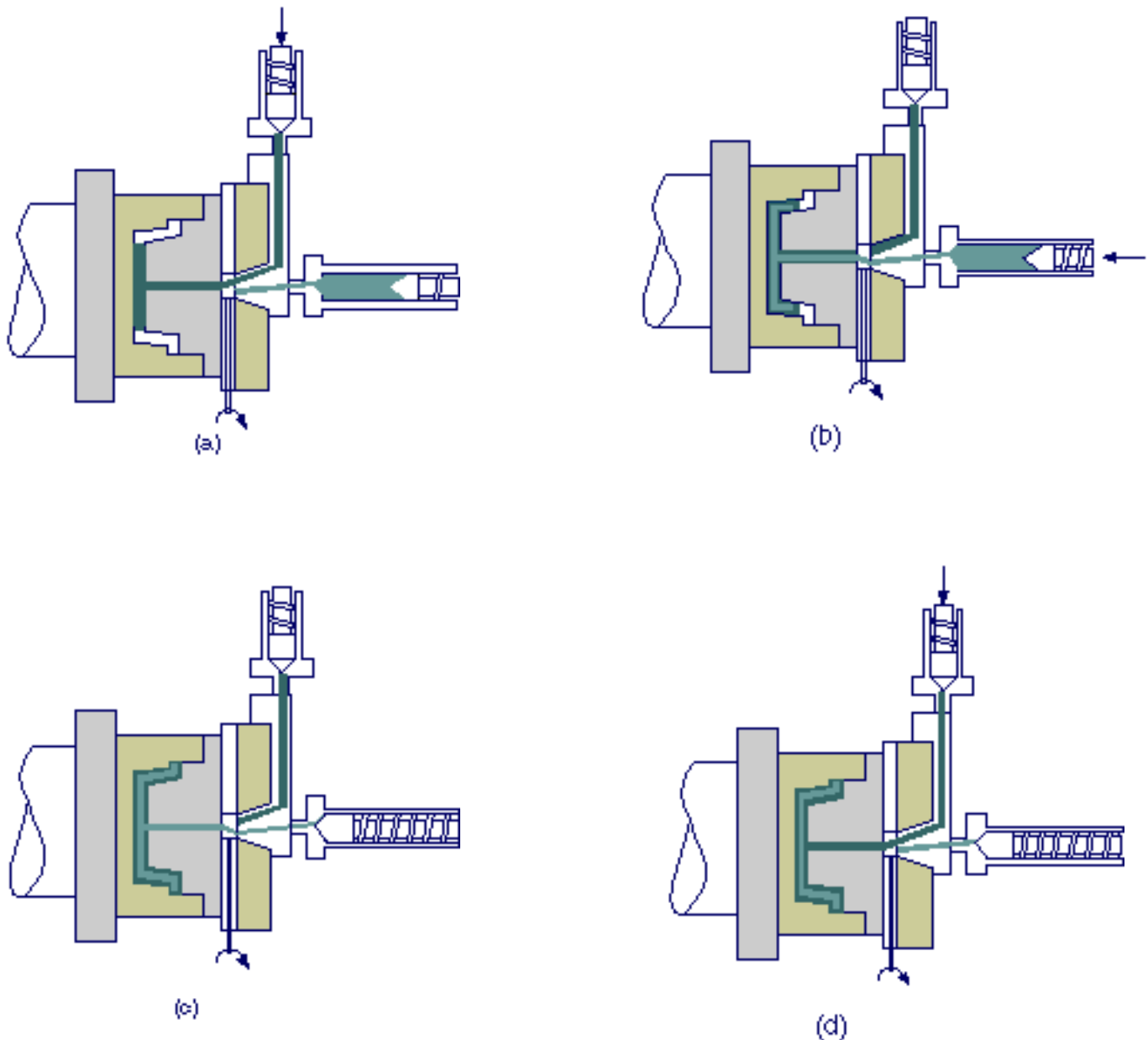


FIGURE 1. Four stages of co-injection molding. (a) Short shot of skin polymer melt (shown in dark green) is injected into the mold. (b) Injection of core polymer melt until cavity is nearly filled, as shown in (c). (d) Skin polymer is injected again, to purge the core polymer away from the sprue.

Fusible core injection molding

Overview

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.

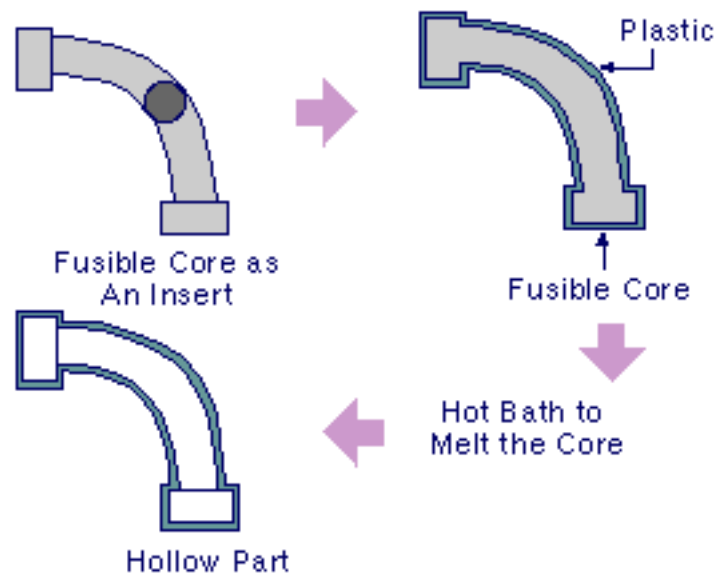


FIGURE 1. Fusible (lost, soluble) core injection molding

Gas-assisted injection molding

Gas-assisted process

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 below.

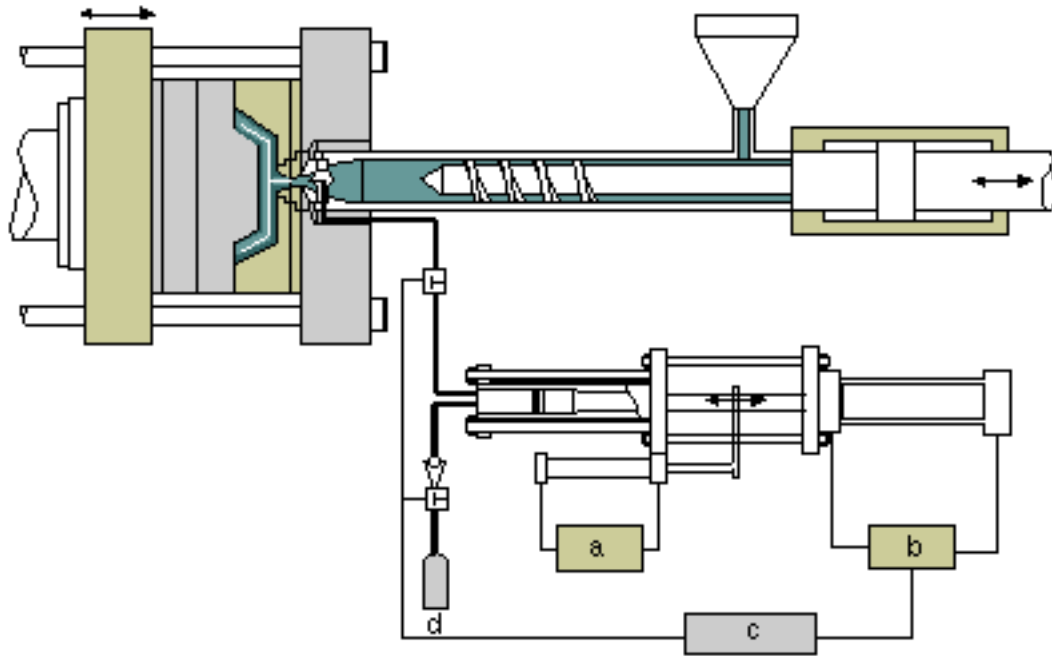


FIGURE 1. Gas-assisted injection molding: (a) the electrical system, (b) the hydraulic system, (c) the control panel, and (d) the gas cylinder.

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.

Typical applications

Typical applications for the gas-assisted injection molding process can be classified into three categories, or some combination of them:

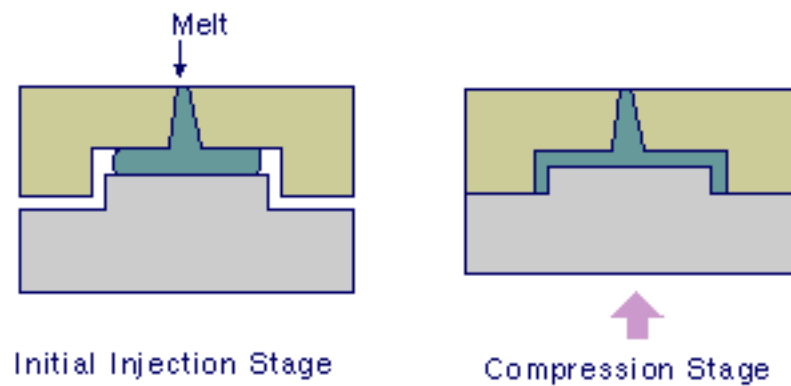
1. Tube-and rod-like parts, where the process is used primarily for saving material, reducing the cycle time by coring out the part, and incorporating the hollowed section with product function. Examples are clothes hangers, grab handles, chair armrests, shower heads, and water faucet spouts.

2. 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.
3. 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.

Injection-compression molding

Overview

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 below. 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 percent lower).



Injection-compression molding

Lamellar (microlayer) injection molding

Overview

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 1](#) below. 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.

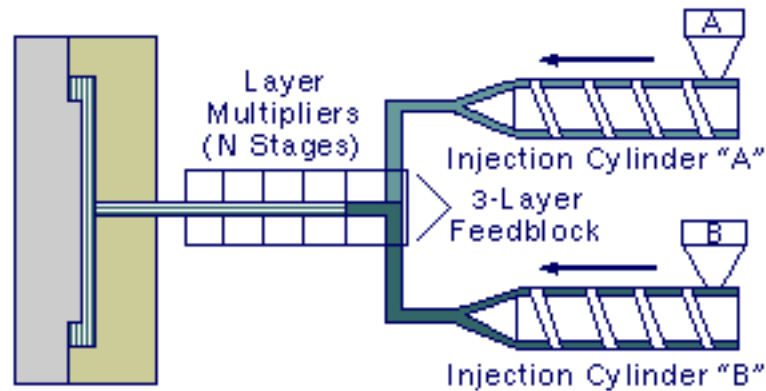
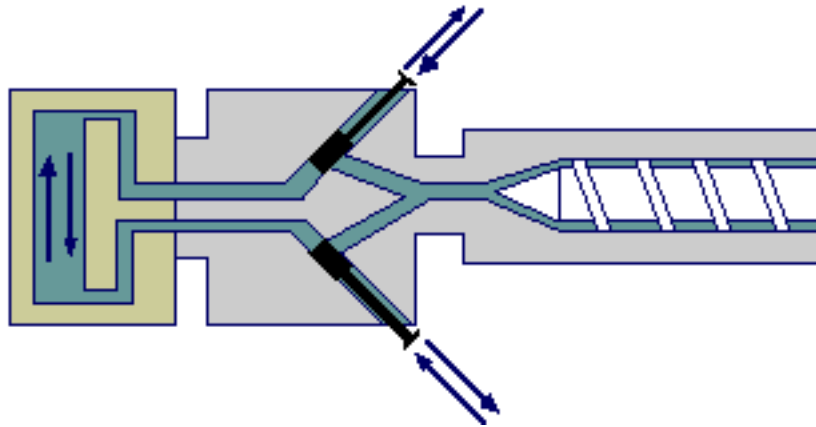


FIGURE 1. Lamellar (microlayer) injection molding

Live-feed injection molding

Overview

The live-feed injection molding process applies oscillating pressure at multiple polymer entrances to cause the melt to oscillate, as shown in the illustration below. 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 due to solidification. This process provides a means of making simple or complex parts that are free from voids, cracks, sink marks, and weld-line defects.



Live-feed injection molding

Low-pressure injection molding

Overview

Low-pressure injection molding is essentially an optimized extension of conventional injection molding (see [Figure 1](#)). 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.

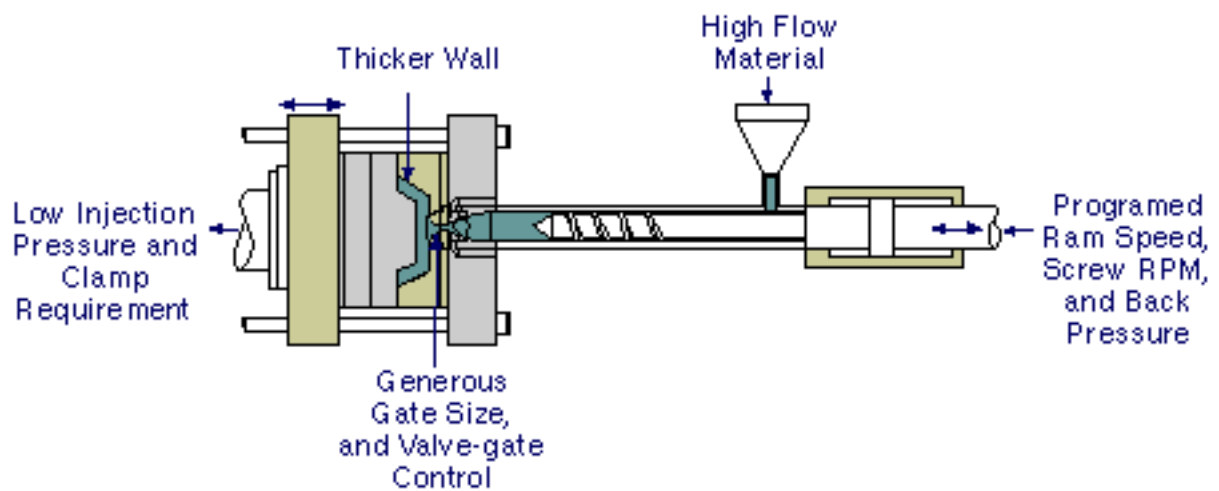


FIGURE 1. Low-pressure injection molding

Push-pull injection molding

Overview

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 below. This process eliminates weld lines, voids, and cracks, and controls the fiber orientation.

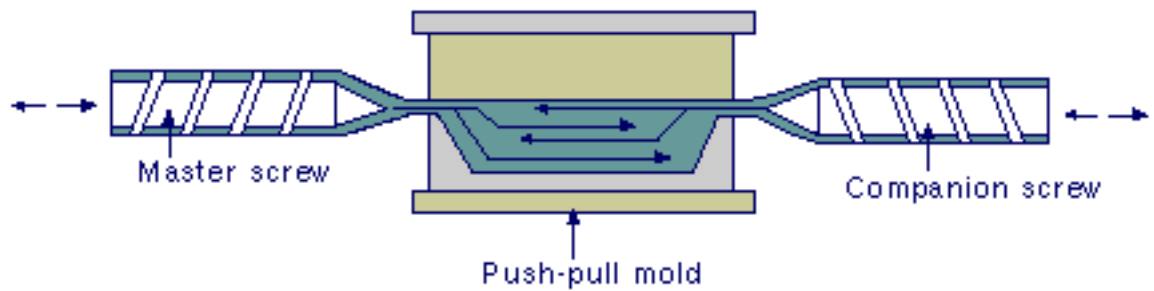


FIGURE 1. Push-pull injection molding

Reactive molding

Overview

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.

Types of reactive materials

Reactive materials include:

- acrylic
- alkyds
- allyl diglycol carbonate
- DAIP
- DAP
- epoxy
- fluorosilicone
- melamine
- melamine/phenolic
- phenolics
- polyurethane polyols
- polyurethane isocyanates
- polyurethane systems
- silicone
- silicone/polymide
- urea
- unsaturated polyesters

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. Another area that is receiving more attention than ever before is the encapsulation of microelectronic IC chips.

The adaptation of injection molding to these materials includes only a small increase in temperature in the feed mechanism (barrel) 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 viscous (frictional) heating of the flow, and from 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.

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 post-filling stages. For instance, slow filling often causes premature gelling and a resultant short shot, while fast filling could induce turbulent flow that creates internal porosity. Improper control of mold-wall temperature and/or inadequate part thickness will either give rise to moldability problems during injection, or cause scorching of the materials. Computer simulation is generally recognized as a more cost-effective tool than the conventional, time-consuming trial-and-error method for tool and process debugging.

Structural foam injection molding

Overview

Structural foam molding produces parts consisting of solid external skin surfaces surrounding an inner cellular (or foam) core, as illustrated in [Figure 1](#) below. This process is suitable for large, thick parts that are subject to bending loads in their end-use application. Structural foam parts can be produced with both low and high pressure, with nitrogen gas or chemical blowing agents.

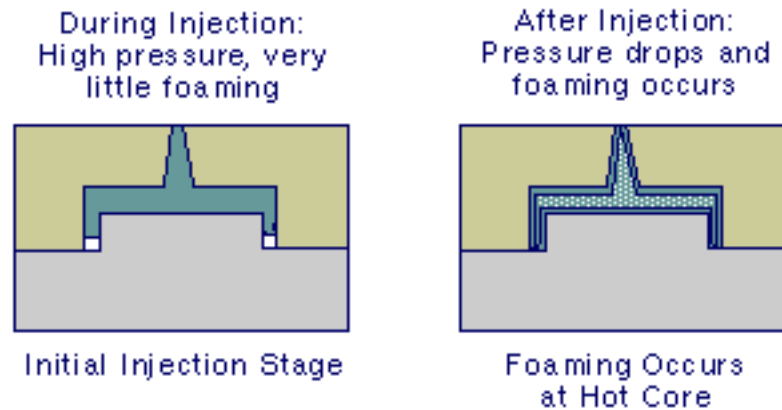


FIGURE 1. Structural foam injection molding

Thin-wall molding

Overview

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.

Typical applications

Thin-wall molding is more popular in portable communication and computing equipment, which demand plastic shells that are much thinner yet still provide the same mechanical strength as conventional parts.

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.

Design

We recommend you design your part 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.

Computer simulation

To accurately simulate thin-wall molding under high pressure, high injection speed, and fast cooling conditions users should specify the following:

- **Pressure dependence of viscosity** to account for the fact that 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 (see [Fast-cooling pVT for semi-crystalline materials](#)).

- **Compression work** to account for the additional heating due to the compression work.

Injection molding machine

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 1](#) below.

- [Injection system](#)
- [Hydraulic system](#)
- [Mold system](#)
- [Clamping system](#)
- [Control system](#)

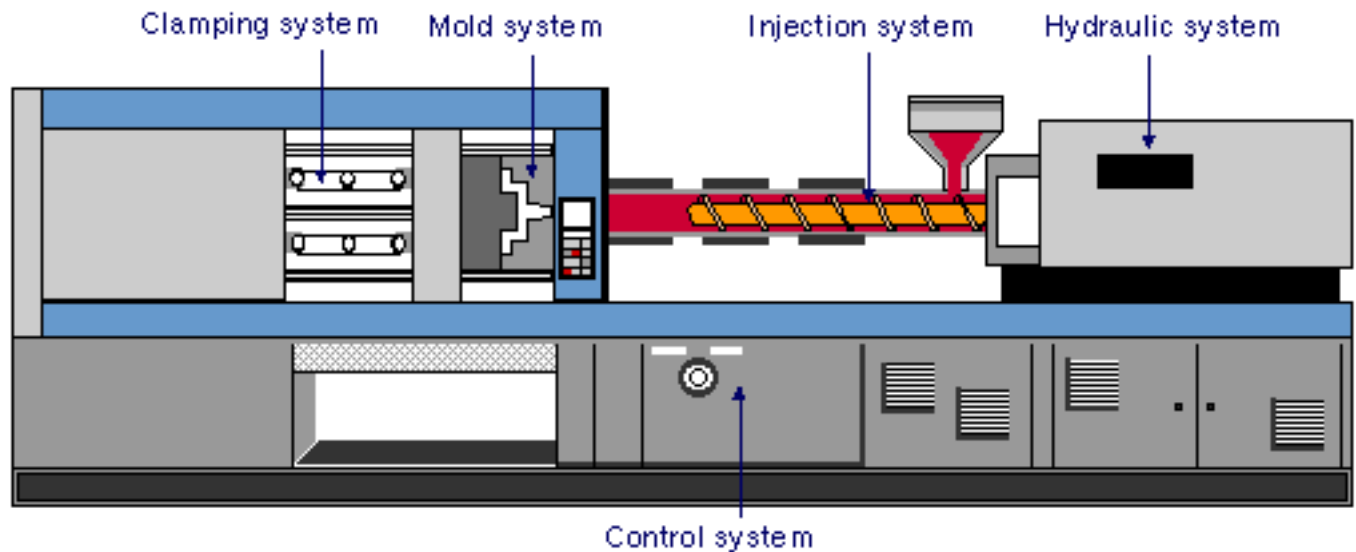


FIGURE 1. A single screw injection molding machine for thermoplastics

Machine specification

Clamping 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.

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

Auxiliary equipment

The major equipment auxiliary to an injection molding machine includes resin dryers, materials-handling equipment, granulators, mold-temperature controllers and chillers, part-removal robots, and part-handling equipment.

Machine operating sequence

Overview

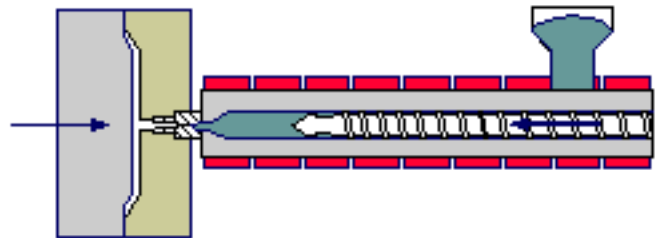
Injection molding is a cyclic process. During the injection molding process, the machine undertakes a sequence of operations in a cyclic fashion. A **process cycle** is one complete operation of an injection molding machine.

Process cycle

The basic injection molding machine operations are shown in the series of diagrams below.

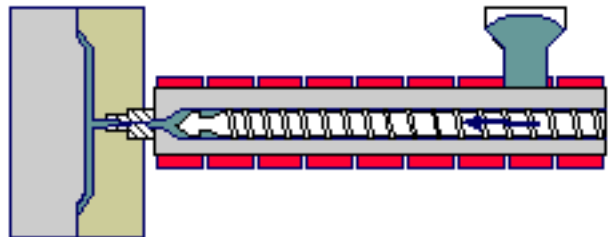
1

The mold closes and the screw begins moving forward for injection.



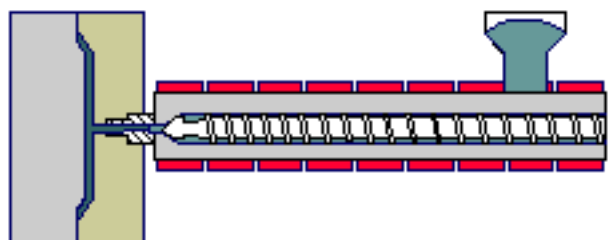
2

The cavity fills as the reciprocating screw moves forward, as a plunger.



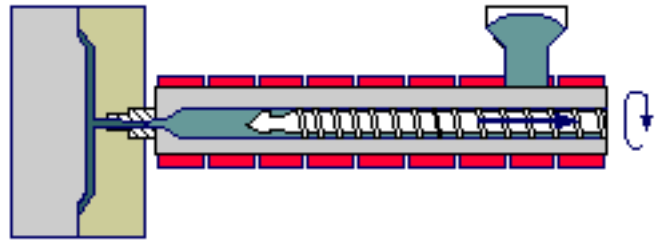
3

The cavity is packed as the screw continuously moves forward.



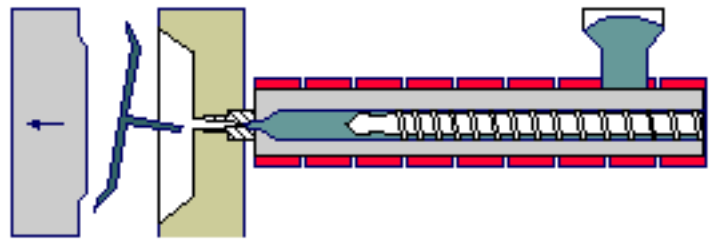
4

The cavity cools as the gate freezes off and the screw begins to retract to plasticize material for the next shot.



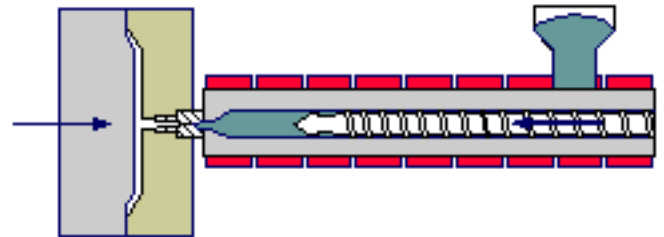
5

The mold opens for part ejection.



6

The mold closes and the next cycle begins



To further illustrate the machine motion within the process cycle, the hydraulic (cylinder) and cavity pressure traces, screw position, and mold face separation position are shown in [Figure 1](#) below. The various stages are:

1 Filling (injection stage)

2 Packing and cooling stage

3 Mold opening

4Part ejection

5Mold closing

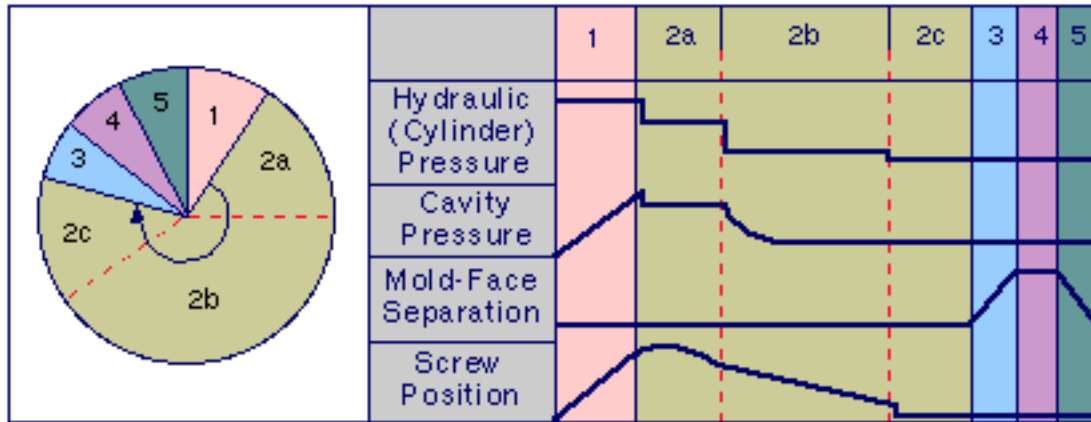


FIGURE 1. A typical injection molding machine cycle clock and the constituent time proportions

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.

Screw operation

Overview

The reciprocating screw is used to plasticize the plastic pellets using various 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 in this document.

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.

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.

Injection speed

The *injection speed* (or ram speed) is the forward speed of the screw during its injection operation.

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 such as the 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.

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.

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 6 mm (1/8 to 1/4 inches) is used.

Secondary operations

Overview

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.

Assembly

Secondary operations for assembling parts include:

- Bonding
- Welding
- Inserting
- Staking
- Swaging
- Assembling with fasteners

Decoration

Secondary operations for decorating the plastic parts include:

- Appliqué: 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.

Other secondary operations

Other secondary operations are:

- Painting
- Hard coating
- Metallizing/shielding
- Surface treatment
- Annealing
- Machining

Setting process conditions

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 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.

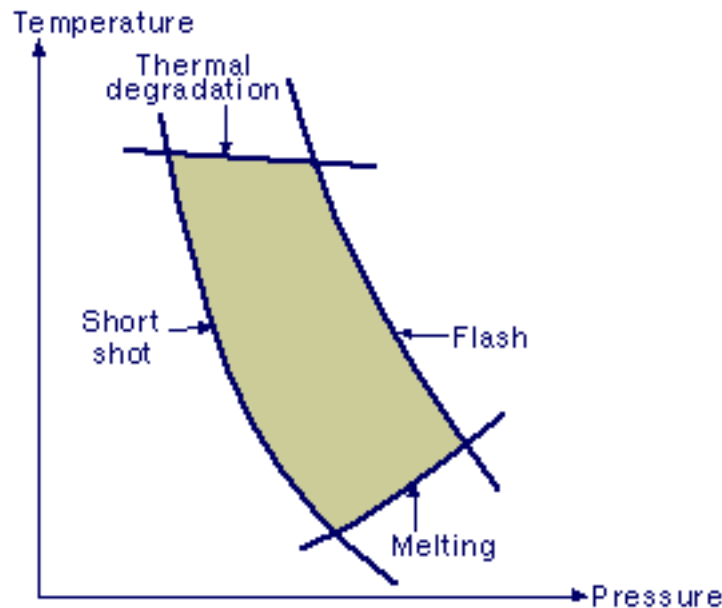


FIGURE 1. Process window shows the influence of pressure versus temperature

After Dr. C-MOLD evaluates the sensitivity of a process to various design parameters, it can produce a machine set-up sheet to assist process engineers in setting injection molding machine controls. See Figure ?. The numbers in the Suggested Value column are parameters optimized by Dr. C-MOLD. In the Set Value column, you may enter the actual machine control settings you used, based on Dr. C-MOLD's values.

Setting machine process conditions

Before setting process conditions, you should make sure 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.

- 1** [Set the melt temperature](#)
- 2** [Set the mold temperature](#)
- 3** [Set the switch-over position](#)

- 4** Set the screw rotation speed
- 5** Set the back pressure
- 6** Set the injection pressure to the machine maximum
- 7** Set the holding pressure at 0 MPa
- 8** Set the injection velocity to the machine maximum
- 9** Set the holding time
- 10** Set ample remaining cooling time
- 11** Set the mold open time
- 12** Mold a short-shot series by increasing injection volume
- 13** Switch to automatic operation
- 14** Set the mold opening stroke
- 15** Set the ejector stroke, start position, and velocity
- 16** Set the injection volume to 99% mold filled
- 17** Increase the holding pressure in steps
- 18** Minimize the holding time

19 Minimize the remaining cooling time **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. Appropriate melt and mold temperatures for several materials are listed in [Resin data table](#). The resin table also contains links to descriptions of resins, their general properties, and typical applications.

► **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 temperature should gradually decrease from the nozzle zone to the zone nearest the hopper.
- The last temperature zone, nearest the hopper, should be about 40° to 50°C (72° to 80°F) 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. Following the rules above, here is an example of how a process engineer would use Dr. C-MOLD's prediction of melt temperature set heater band temperatures. Dr. C-MOLD predicts 235°C (455°F) as the melt temperature for PS, as shown in Figure ?. A process engineer can use this melt temperature to set the heater band temperature as follows:

- 235°C (455°F) at the nozzle zone
- 235°C (455°F) at the front zone
- 210°C (410°F) at the first middle zone
- 195°C (383°F) at the second middle zone
- 180°C (356°F) at the rear zone

► **Air-shot temperature**

The actual melt temperature, or air-shot temperature, is usually higher than the heater band controller setting. This difference is due to 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.)

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 [Resin data table](#).

The mold temperature can be measured by using a thermometer. As illustrated below, 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 20°C (18° to 36°F) lower than the required mold temperature (provided on Dr. C-MOLD's setup sheet). If the mold temperature is 40° to 50°C (72° to 80°F) or more, consider insulation plates between the mold and the clamping plates, for energy savings and process stabilization.

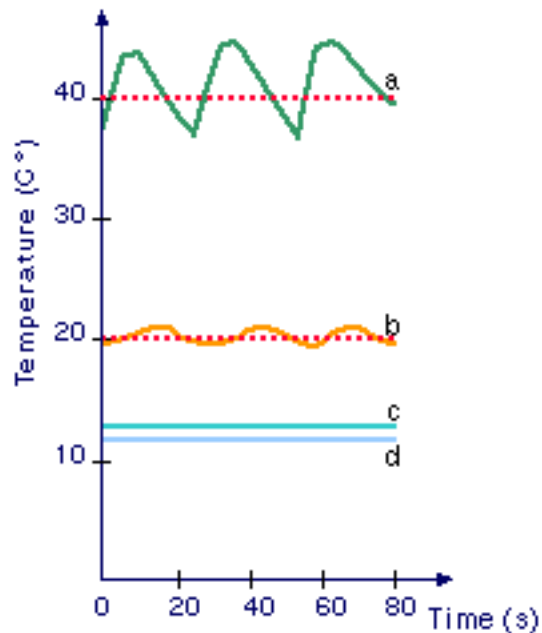


FIGURE 2. Temperature-time curve at various locations in the mold. a) Mold cavity surface. b) Cooling channel wall. c) Cooling channel outlet. d) Cooling channel inlet.

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.

Step 3

Set the switch-over position

The switch-over position is the ram position where the filling (injection) stage switches to the post-filling (packing or holding) stage. The cushion distance is the distance from the switch-over position to the farthest position that the end of the screw can reach, as shown in [Figure 3](#). Thus, the switch-over position determines the cushion distance. The cushion should contain adequate material for post-filling the part. An insufficient cushion could cause sink marks. The typical cushion distance is about 5 to 10 mm.

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 12](#), the injection volume will be increased to fill the cavity.

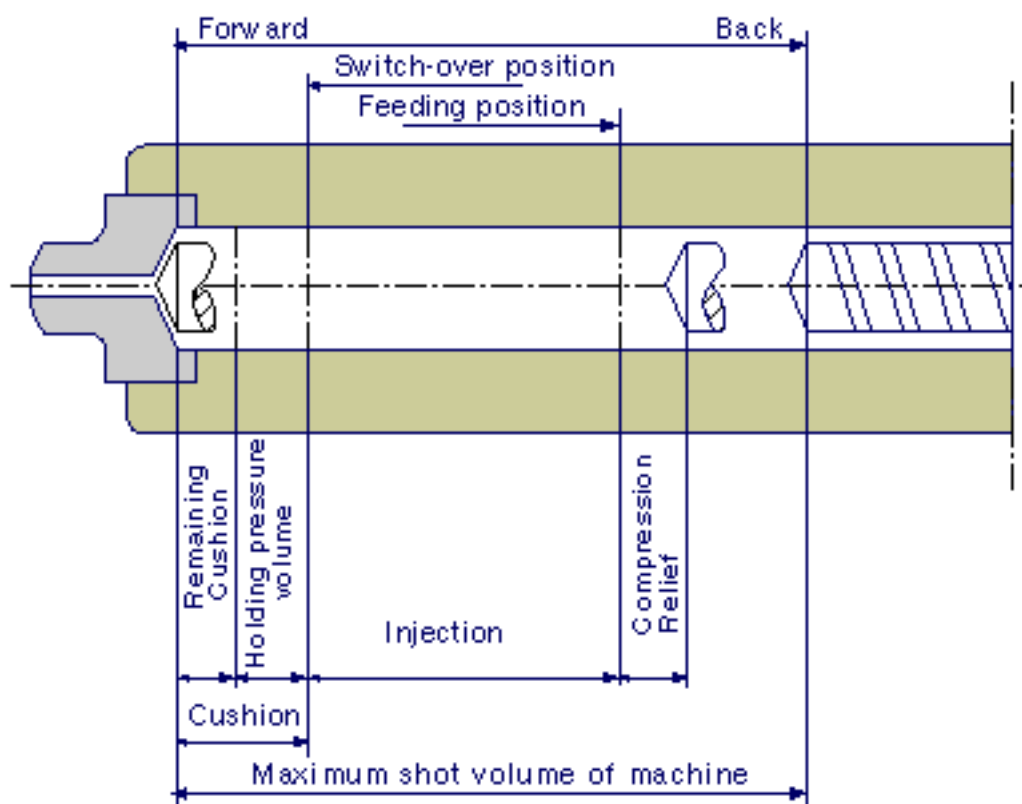


FIGURE 3. Screw positions at various stages

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 prolonging the cycle time. Resin vendors supply the suggested screw rotation speed for specific resins.

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. Use a higher back pressure to achieve a shot volume that is a larger percentage of the injection machine's capacity, in order to speed up plasticization. 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.

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 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 holding pressure occurs before the mold is completely filled, no damage will be done to the mold.

Step 7

Set the holding pressure at 0 MPa

For now, set the [holding 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 holding pressure is increased to its final setting.

Step 8

Set the injection velocity to the machine maximum

With the highest possible [injection velocity](#), you can expect less flow resistance, longer flow length, and improved strength in weld lines. However, you may need to create 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. C-MOLD shows you where [weld lines](#), [meld lines](#), and air trap locations will occur; use these predictions improve your design. Remember that it is necessary to clean the mold surface and venting system regularly, especially for PVC or ABS/PVC materials.

The filling time estimated by Dr. C-MOLD is based on the maximum injection velocity for the selected machine size, part volume, and the injection pressure. A higher injection pressure requirement will slow the injection velocity, thus resulting in a longer filling time.

The actual filling time on the shop floor may be shorter if there is a booster or accumulator attached to the injection unit, or may be longer if the injection velocity is not set to the maximum. Also note that the shop floor filling time often refers to "the time while the screw is moving," which includes filling time *and* holding time. The actual filling time should stop at the switch-over position.

Step 9

Set the holding time

The ideal [holding 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. For your first trials, you can estimate the holding time to be 10 times the filling time predicted by C-MOLD.

Dr. C-MOLD calculates the holding time for your design. The holding time calculated by Dr. C-MOLD is the gate freeze time or the part freeze time, whichever is shorter. Use this as your initial holding time setting. It can be further refined in Step 18.

Step 10

Set ample remaining cooling time

[Cooling time](#) can be calculated or estimated. The cooling time consists of the holding time and the remaining cooling time, as shown in [Figure 4](#). Your first estimate of the cooling time can be 10 times the filling time. For example, if the predicted filling time is .85 seconds, the initial holding time would be 8.5 seconds and the additional cooling time would be 8.5 seconds. This ensures that the part and runner system will be sufficiently solid for ejection.

Cooling time is estimated by Dr. C-MOLD and included on the machine setup sheet. The cooling time consists of the holding time and the remaining cooling time. See Figure ?. Dr. C-MOLD estimates the cooling times for the cavity. In practice, a part may have to wait until the runner is solidified before ejection. Use the estimated cooling time from Dr. C-MOLD first. Increase the cooling time if needed.

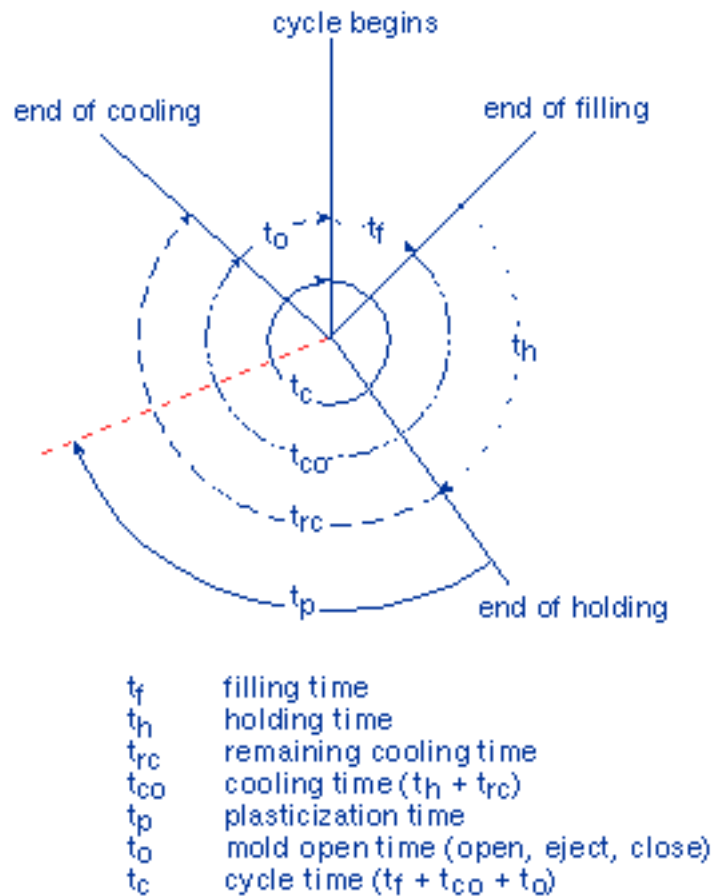


FIGURE 4. Cycle time and its components

Step 11

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 4](#). The cycle time is the sum of the filling time, cooling time, and mold open time.

Step 12

Mold a short-shot series by increasing injection volume

C-MOLD 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 [Screw positions at various stages](#)) can be estimated for each shot.

For now, fill only two-thirds of the mold. The holding 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 percent, up to 95 percent 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.

Step 13

Switch to automatic operation

The purpose of an automatic operation is to obtain stability in the process.

Step 14

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 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.

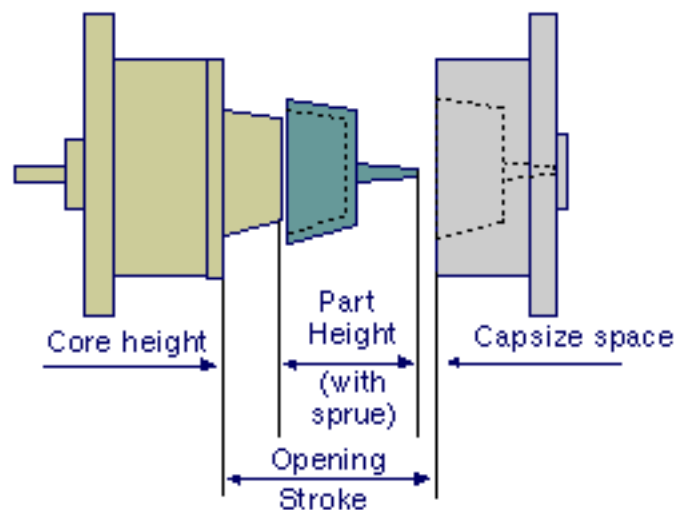


FIGURE 5. Required mold opening

Step 15

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.)

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 percent of filling. This will exploit the maximum injection speed in as

large a part of the injection as possible.

Step 17

Increase the holding pressure in steps

Increase the [holding 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 holding pressure on it. This holding pressure series forms a good basis for a more thorough examination. You can then discuss the possibilities and limitations with the customer.

Dr. C-MOLD provides a holding pressure that results in minimum shrinkage, without flashing or demolding problems. This estimated holding pressure is typically the maximum value, and you should not set holding pressure higher than the value provided by Dr. C-MOLD.

Choose the lowest acceptable holding pressure, as this minimizes the internal stresses in the part and saves material, as well as operating costs. A high holding 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 [Screw positions at various stages](#)), the last part of the holding 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 hydraulic pressure by the resin/hydraulic pressure ratio. (See [Factors that influence injection pressure requirements](#).) 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 below.

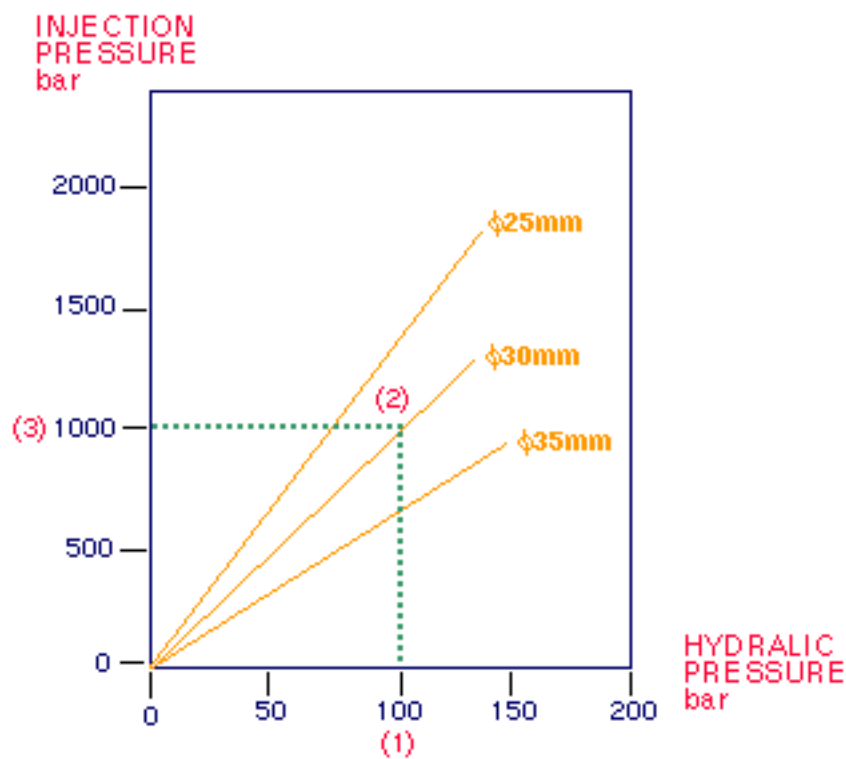


FIGURE 6. Resin/hydraulic pressure ratio for a Ø 30 mm screw is 11.1

Step 18

Minimize the holding time

A quick way to find the minimum holding time is by setting a longer holding time, then reducing the holding time until sink marks appear.

If consistent part dimensions are essential, use the following more accurate determination of the holding pressure time. From a curve of part weight versus holding time, determine when the gate or the part freezes. For example, [Figure 7](#) shows that the holding pressure does not influence the part weight after nine seconds. This is your minimum holding time.

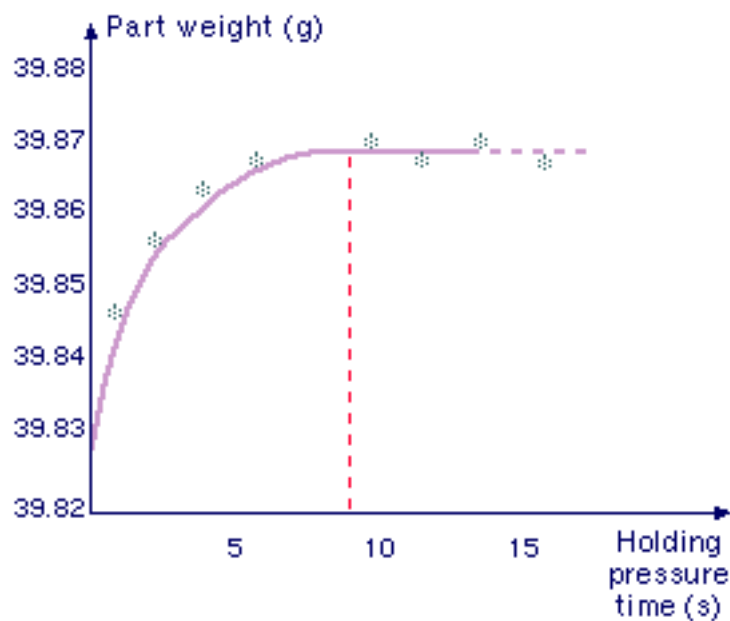


FIGURE 7. Determination of the gate/part freezing time by weighing parts manufactured at various holding pressure times

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.

What are plastics?

Polymerization process

Plastics are one group of polymers that are built from relatively simple units called monomers (or mers) through a chemical polymerization process. This process is illustrated below. Processing polymers into end products mainly involves physical phase change such as melting and solidification (for [Thermoplastics](#)) or a chemical reaction (for [Thermosets](#)).

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 1](#). Although the term plastics has been used loosely 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 processability 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 below.

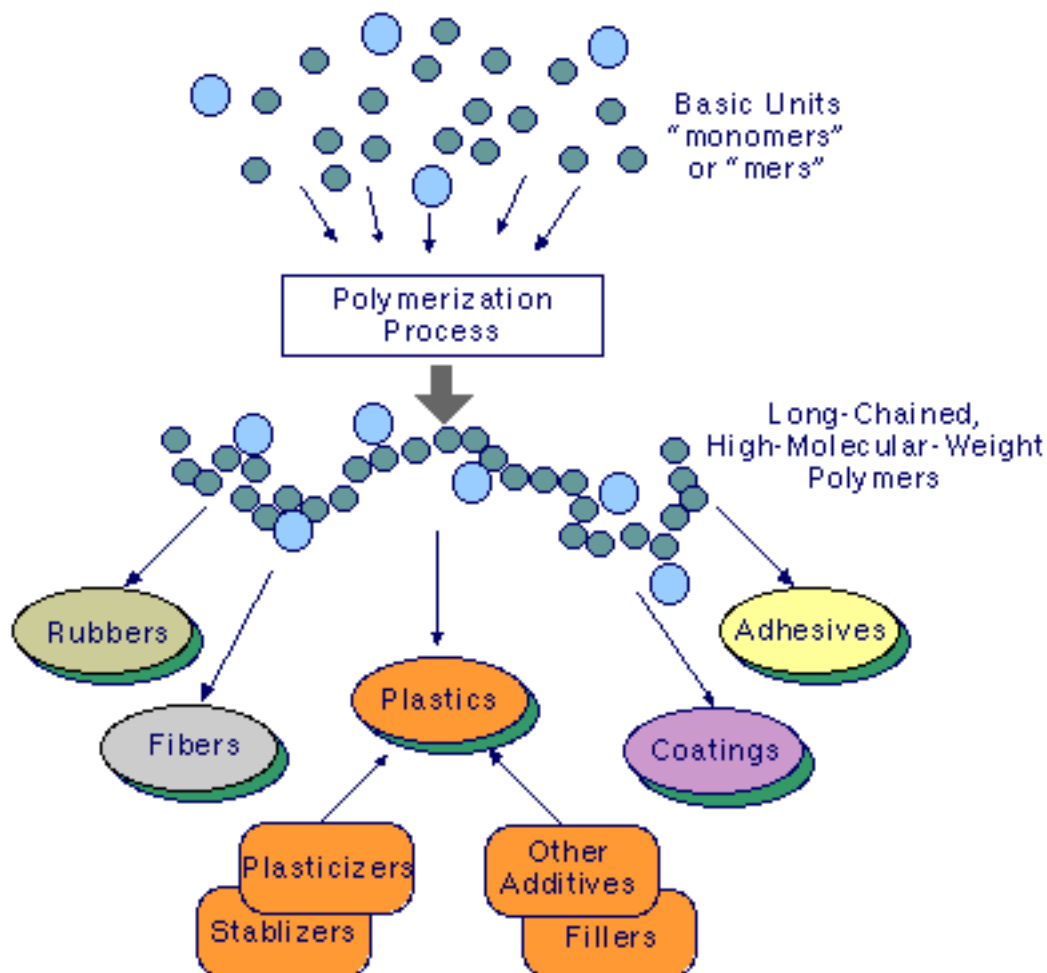
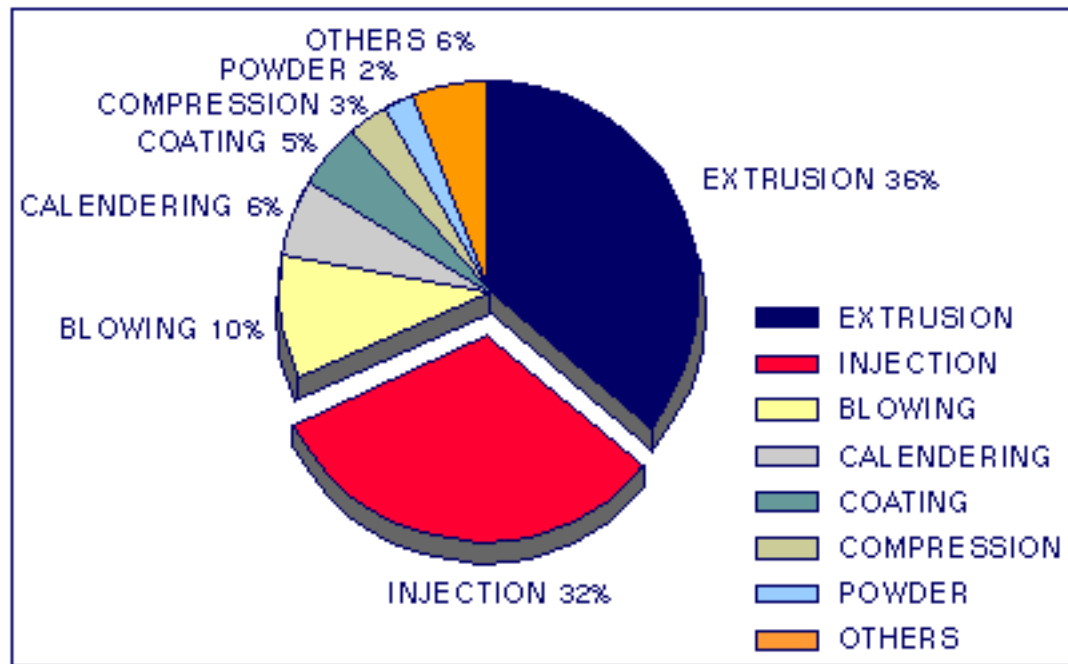


FIGURE 1. Polymer family, the formation of plastics, and the polymerization process**FIGURE 2.** Plastics consumption by process

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.

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.

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.

Classification of plastics

Based on chemical reaction

Based on the type of chemical reaction (polymerization) that links the molecules together, plastics are classified as either thermoplastics or thermosets.

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 1](#). 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.

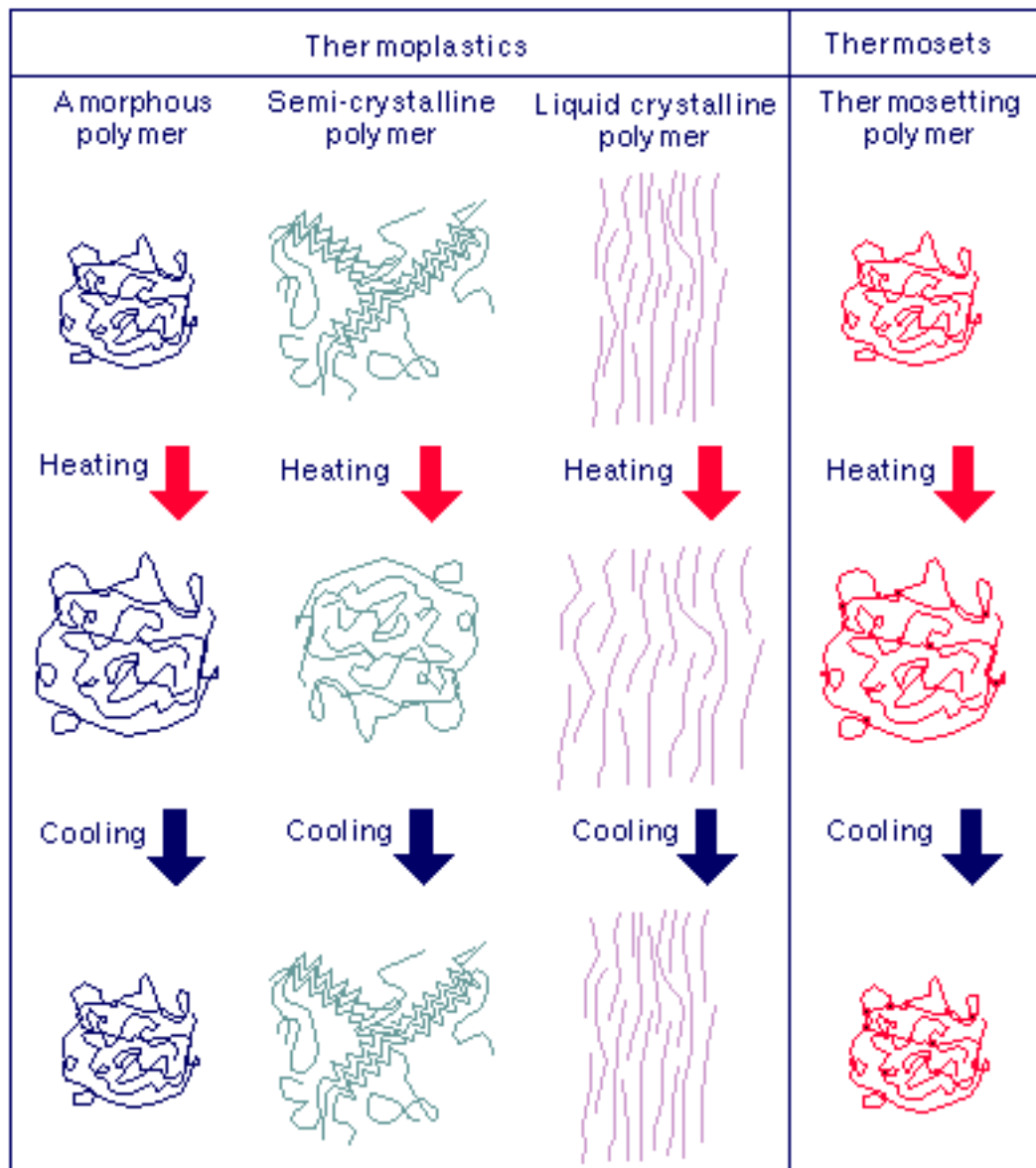


FIGURE 1. Microstructure of various plastics and effect of heating and cooling during processing

Structures and properties of plastics

Table 1 lists a summary of the relevant structures and properties of thermoplastics and thermosets.

TABLE 1. Structures and properties of thermoplastics and thermosets

Thermoplastics	Thermosets
Microstructure	
<ul style="list-style-type: none">■ Linear or branch molecules.■ No chemical bonds among the molecules.	<ul style="list-style-type: none">■ Cross-linking network with chemical bonds among molecules after the chemical reaction.
Reaction to Heat	
<ul style="list-style-type: none">■ Can be re-softened (physical phase change).	<ul style="list-style-type: none">■ Cannot be re-softened after cross-linking without degradation.
General Properties	
<ul style="list-style-type: none">■ Higher impact strength.■ Easier processing.■ Better adaptability to complex designs.	<ul style="list-style-type: none">■ Greater mechanical strength.■ Greater dimensional stability.■ Better heat and moisture resistance.

Thermoplastics

Usefulness of 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, thermal degradation) takes place during processing, and therefore the properties of recycled polymers may not be equivalent to those of the virgin polymer.

Market share distribution of thermoplastics

Thermoplastics account for more than 70 percent 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.

► **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 percent of all thermoplastics.

► **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, higher impact strength. Thus, they demand a higher price.

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 temperature (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)

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 [Microstructure of various plastics and effect of heating and cooling during processing](#).

► 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.

► 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.

► 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.

(Semi-)crystalline 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 [Microstructure of various plastics and effect of heating and cooling during processing](#).

► 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 percent crystallization under normal processing conditions, any crystallizable polymers are typically semi-crystalline, possessing both amorphous and crystalline phases. The degree of crystallinity depends on both the chemical structure of the polymer and the processing conditions. [Fast-cooling pVT for semi-crystalline materials](#) discusses how a fast cooling rate, which occurs during molding, affects the material's transition to the crystalline phase. (Semi-)crystalline polymers have a distinct melting point, good chemical and heat resistance, good lubricity, low moisture absorption, and high shrinkage.

► High linear shrinkage

The significantly higher linear shrinkage of the semi-crystalline 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.

Liquid crystal polymers

Liquid crystal polymers (LCPs) exhibit ordered molecular arrangements in both the melt and solid states, as shown in [Microstructure of various plastics and effect of heating and cooling during processing](#). These materials are characterized by their stiff, rod-like 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.

Thermosets

How they differ from thermoplastics

Cross-linking is a chemical process in which chemical bonds form among molecules of thermosetting materials, resulting in an interconnected network, as shown in [Microstructure of various plastics and effect of heating and cooling during processing](#).

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.

Cross-linking (reaction)

Prior to molding, the chain-like 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.

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.

Additives, fillers, and reinforcements

Why they're important

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 1. Effects of additives, fillers, and reinforcements on polymer properties

Additive / Filler / Reinforcement	Common materials	Effects on polymer properties
Reinforcing fibers	Baron, carbon, fibrous minerals, glass, Kevlar	<div><div></div>Increases tensile strength.</div> <div><div></div>Increases flexural modulus.</div> <div><div></div>Increases heat-deflection temperature (HDT).</div> <div><div></div>Resists shrinkage and warpage.</div>

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. Anti-static 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.

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.

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.

► 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.

► 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.

► 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, in order to predict the properties of the molded article.

Resin data table

Clicking on a resin in the table will take you to a more detailed description of that resin.

Generic Name	Flow Properties			Melt Temperature (C/F)			Mold Temperature (C/F)			Ejection Temp. (C/F)
	MFR g/10min	Test Load kg	Test Temp. C	Min.	Rec.	Max.	Min.	Rec.	Max.	Recommended
ABS	35	10	220	200/392	230/446	280/536	25/77	50/122	80/176	88/190
PA 12	95	5	275	230/446	255/491	300/572	30/86	80/176	110/230	135/275
PA 6	110	5	275	230/446	255/491	300/572	70/158	85/185	110/230	133/271
PA 66	100	5	275	260/500	280/536	320/608	70/158	80/176	110/230	158/316
PBT	35	2.16	250	220/428	250/482	280/536	15/60	60/140	80/176	125/257
PC	20	1.2	300	260/500	305/581	340/644	70/158	95/203	120/248	127/261
PC ABS	12	5	240	230/446	265/509	300/572	50/122	75/167	100/212	117/243
PC PBT	46	5	275	250/482	265/509	280/536	40/104	60/140	85/185	125/257
PE-HD	15	2.16	190	180/356	220/428	280/536	20/68	40/104	95/203	100/212
PE-LD	10	2.16	190	180/356	220/428	280/536	20/68	40/104	70/158	80/176
PEI	15	5.00	340	340/644	400/752	440/824	70/158	140/284	175/347	191/376
PET	27	5	290	265/509	270/518	290/554	80/176	100/212	120/248	150/302
PETG	23	5	260	220/428	255/491	290/554	10/50	15/60	30/86	59/137
PMMA	10	3.8	230	240/464	250/482	280/536	35/90	60/140	80/176	85/185
POM	20	2.16	190	180/356	225/437	235/455	50/122	70/158	105/221	118/244
PP	20	2.16	230	200/392	230/446	280/536	20/68	50/122	80/176	93/199
PPE PPO	40	10	265	240/464	280/536	320/608	60/140	80/176	110/230	128/262
PS	15	5	200	180/356	230/446	280/536	20/68	50/122	70/158	80/176
PVC	50	10	200	160/320	190/374	220/428	20/68	40/104	70/158	75/167
SAN	30	10	220	200/392	230/446	270/518	40/104	60/140	80/176	85/185

How does plastic flow?

Material behavior

Molten thermoplastic exhibits 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.

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 (a) and (b) below. The flow of molten thermoplastics during injection molding filling is predominantly shear flow, as shown in (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 (d).

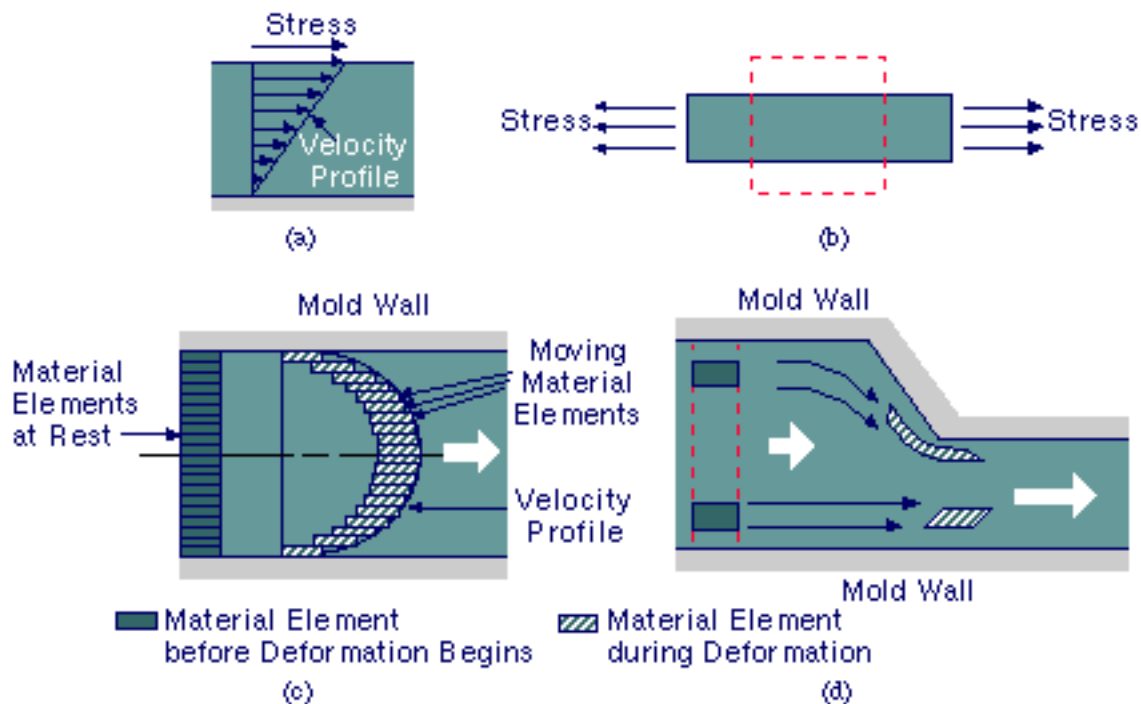


FIGURE 1. (a) Simple shear flow. (b) Simple extensional flow. (c) Shear flow in cavity filling. (d) Extensional flow in cavity filling.

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 below. Upon the removal of the stress, however, the materials behave somewhat like an elastic solid with partial recovery of the deformation, as shown in (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.

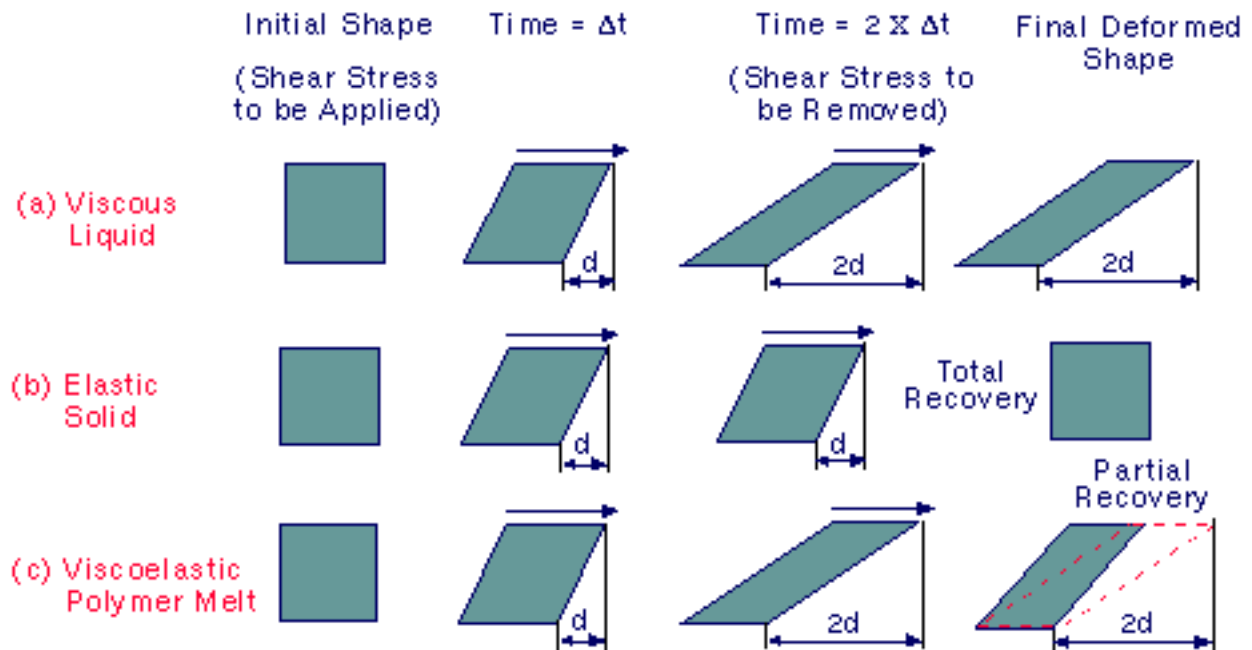


FIGURE 2. (a) Ideal viscous liquid deforms continuously under applied stress. (b) Ideal elastic solid deforms immediately upon the application of stress, but fully recovers when the stress is removed. (c) Molten thermoplastic deforms continuously under the applied stress (like a viscous liquid), but it also recovers partially from the deformation upon removal of the applied stress (like an elastic solid).

Melt shear viscosity

What is shear viscosity?

Melt shear viscosity is a material's resistance to shear flow. In general, polymer melts are highly viscous due to their long molecular chain structure. The viscosity of polymer melt ranges from 2 to 3,000 Pas (water 10^{-1} , glass 10^{20}). 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 the equation and diagram below:

$$\text{Viscosity} = \frac{\text{Shear Stress}}{\text{Shear Rate}}$$

where

$$\text{Shear Stress} = \frac{\text{Force (F)}}{\text{Area (A)}} \text{ and Shear Rate} = \frac{\text{Velocity (v)}}{\text{Height (h)}}$$

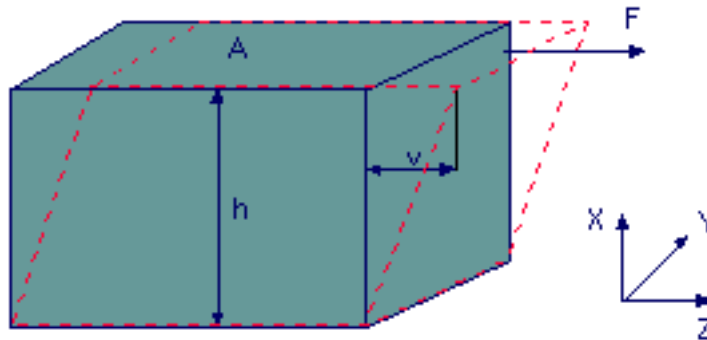


FIGURE 1. The definition of polymer melt viscosity, illustrated by a simple shear flow

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.

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, the resistance exhibited by polymer to flow decreases with the deformation, 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, since the water does not have shear-thinning behavior. But in a similar situation using a polymer melt, if the pressure is doubled, the melt flow rate may increase from 2 to 15 times, depending on the material.

Shear rate distribution

Having introduced the concept of shear viscosity, let us look at the shear rate distribution in the cavity during injection molding. Generally speaking, 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 (a), it is clear that the shear rate is highest at the mold-melt interface (or at the melt-solid interface if there is a frozen polymer layer). On the other hand, the shear rate approaches zero at the center line because there is no relative material element movement due to flow symmetry, as shown in [Figure 2](#) (b). Shear rate is an important flow parameter since it influences the melt viscosity and the amount of shear (viscous) heating. The typical shear rate experienced by the polymer melt during the injection molding process ranges from 10^2 to 10^5 second⁻¹.

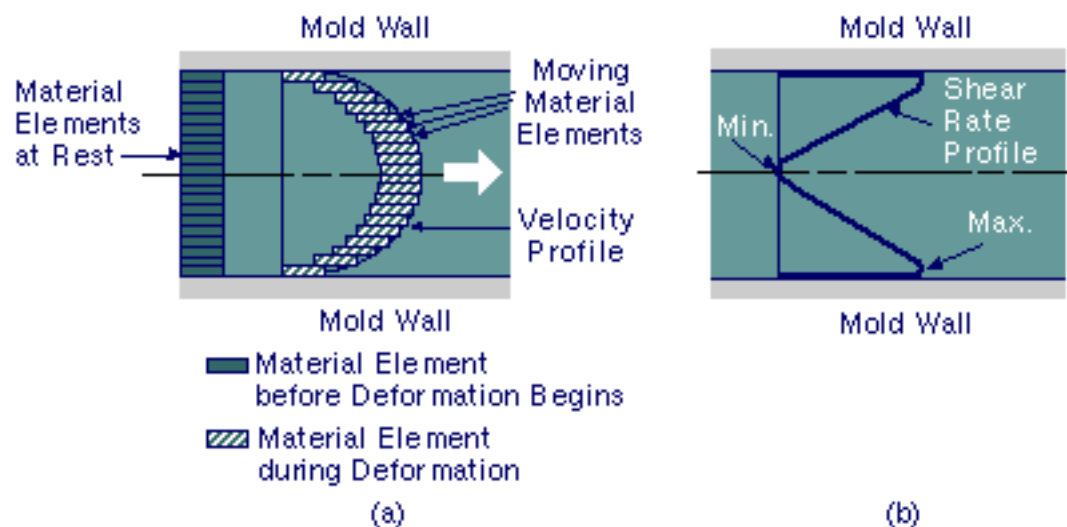


FIGURE 2. (a) A typical velocity profile with relative flow element movement and (b) the corresponding shear rate distribution in injection molding filling.

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 3](#), the melt viscosity decreases with increasing shear rate and temperature due to 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.

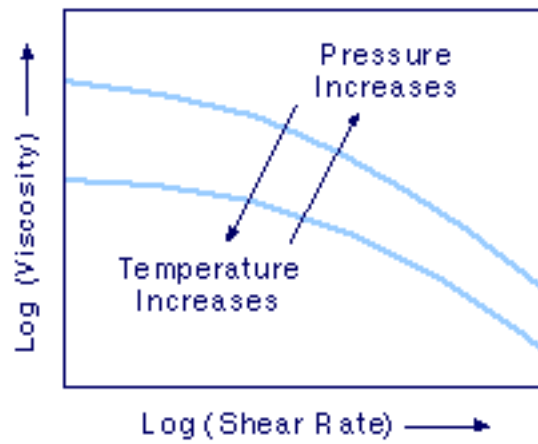


FIGURE 3. The viscosity of polymer melt depends on the shear rate, pressure, and temperature.

Rheological material properties contains a mathematical description of the shear viscosity as a function of shear rate, temperature, and pressure. For a discussion on how high pressure increases the level of viscosity, see Pressure dependence of viscosity.

Pressure-driven flow

Overview

Flow of molten thermoplastics (in injection molding filling) is driven by pressure that overcomes the melt's resistance to flow. Molten thermoplastics flow from high pressure areas to the low pressure areas, 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 gradually decreases along the flow length toward the polymer melt front, where the pressure reaches the atmospheric pressure, if the cavity is vented properly.

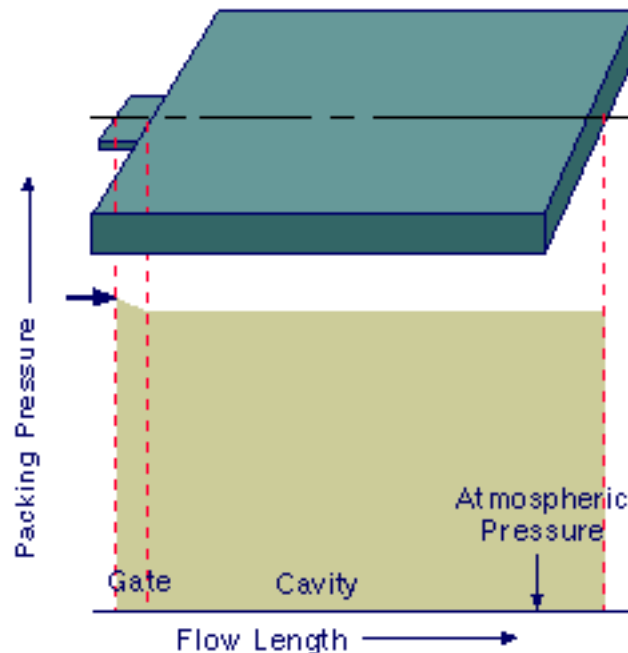


FIGURE 1. Evolution of pressure distribution within the cavity during filling and early packing (Click [Replay](#))

Pressure gradient and melt speed

The higher the pressure and resultant pressure gradient (pressure drop per unit flow length) at the melt entrances, the faster the material flows. Therefore, increased flow length requires increased entrance pressure, in order to generate the same pressure gradient to maintain the polymer melt speed, as shown in [Figure 2](#) below.

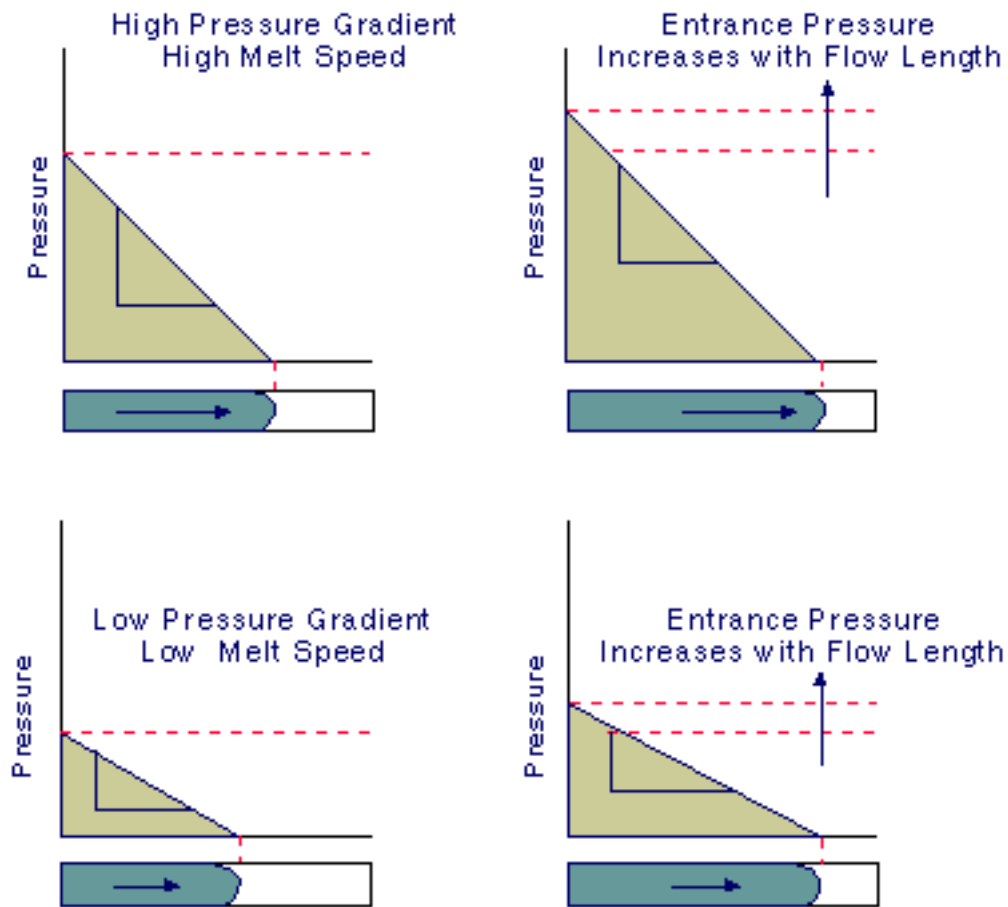


FIGURE 2. The relationship of melt velocity to the pressure gradient.

Flow conductance

The speed of the melt also depends on the flow conductance, an index of how easily the melt can flow. Flow conductance, in turn, is a function of the geometry (e.g., wall thickness, surface features) and the melt viscosity. The flow conductance increases with increasing wall thickness and decreases with increasing melt viscosity, as shown below.

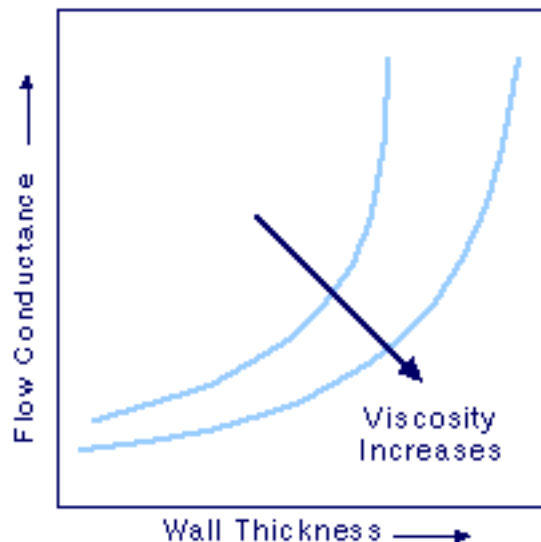


FIGURE 3. The relationship of flow conductance to the wall thickness and viscosity.

Melt flow length

During injection molding, the distance that the material can flow, with certain processing conditions and wall thickness, is dependent on the thermal properties and shear properties of the material. This behavior can be characterized by the melt flow length, as illustrated in [Figure 4](#) below.

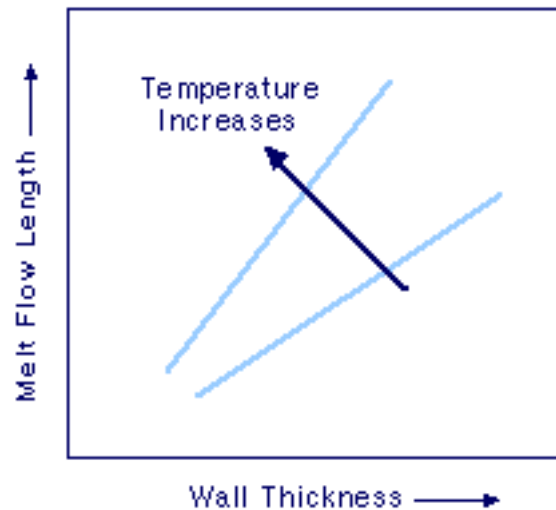


FIGURE 4. The melt flow length depends on the part thickness and temperature.

Injection pressure vs. fill time

For injection molding, if the injection pressure required to fill the cavity is plotted against the fill time, a U-shaped curve typically results, with the minimum value of the required injection pressure occurring at an intermediate fill time, as illustrated below. The curve is U-shaped because, on the one hand, a short fill time involves a high melt velocity and thus requires a higher injection pressure to fill the mold. On the other hand, the injected polymer cools more with a prolonged fill time. This results in a higher melt viscosity and thus requires a higher injection pressure to fill the mold. The shape of the curve of injection pressure versus fill time depends very much on the material used, as well as on the cavity geometry and mold design.

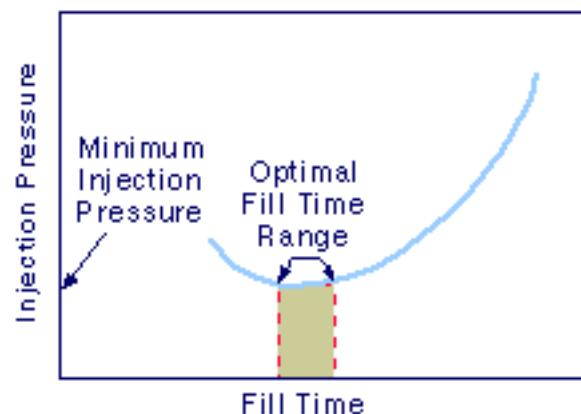


FIGURE 5. U-shaped curve of injection pressure vs. fill time.

Flow instability

Finally, it should be pointed out that 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.

Fountain flow

Overview

The phenomenon of **fountain flow** or **fountain effect** refers to a simple shear flow behind the melt front that is transformed into a combination of shear and extensional flows as hot fluid elements from the core area move toward the mold wall and subsequently solidify.

Effect of fountain flow

The dynamics of fountain flow are becoming better understood, with the increasing publication of data on polymer melts with tracers, and the growing use of video tape of the experiments. The material that enters the cavity earlier shows up at the surface close to the gate; the material that enters the cavity later shows up on the part surface downstream. Because the material at the melt front turning toward the mold surface from the hot core solidifies immediately, fountain flow has a direct influence on the molecular and fiber orientation at the part surface. This is why filling the cavity with a constant [melt-front velocity](#) leads to uniform surface orientation and better surface quality.

ABS (Acrylonitrile-Butadiene-Styrene)

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, lawnmowers, etc.)
- Telephone housings, typewriter housings, typewriter keys
- Recreational vehicles such as golf carts and jet skis.

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:	210 - 280 C (410 - 536 F); Aim: 245 C (473 F)
Mold Temperature:	25 - 70 C (77 - 158 F). (Mold temperatures control the gloss properties; lower mold temperatures produce lower gloss levels)
Injection Pressure:	500 - 1,000 bar (7,250 - 14,500 psi)
Injection Speed:	Moderate - high

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; processibility, 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, which 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 processibility, appearance, low creep and excellent dimensional stability, and high impact strength.

Major Manufacturers

- Dow Chemical (Magnum grades)
- GE Plastics (Cycolac)
- Bayer (Lustran)
- BASF (Terluran)
- Chi Mei (Polylac)
- LG Chemical (Lupos)
- Cheil Synthesis.

PA 12 (Polyamide 12 or Nylon 12)

Typical Applications

- Gear wheels for water meters and business machines
- Cable ties
- Cams
- Slides
- Bearings

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 -5 hours is recommended (3-4 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:	240 - 300 C (464 - 580 F); Not to exceed 310 C (590 F) for standard grades and 270 C (518 F) for flame retardant grades
Mold Temperature:	30 - 40C (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 1,000 bar (14, 500 psi) Low hold pressures and high melt temperatures are recommended.
Injection Speed:	High (high speeds give better finish on glass-filled grades)

Runners and Gates

Runner diameters for unfilled grades may be as small as 3 - 5 mm because of the material's low viscosity. Reinforced grades require larger diameters (5 - 8 mm). The runner shape should be the full round type. Sprues should be as short as possible.

A variety of gates may be used. Small gates for large parts should be not be used, in order to avoid highly stressed components or excessive shrinkage. The thickness of the gate should preferably 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.

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 resistant to 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, density, and 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 - 0.02 mm/mm (0.5 - 2%). This is dependent on the specific grade, wall thickness, and processing conditions.

Major Manufacturers

- Huls (Vestamid)
- Atochem (Rislon A).

PA 6 (Polyamide 6, Nylon 6, or Polycaprolactam)

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.

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 - 280 C (482 - 536 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-wall 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 750 - 1,250 bar (~11,000 - 18,000 psi) (depends on material and product design)
Injection Speed:	High (slightly lower for reinforced grades)

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 $0.5 \cdot t$ (where "t" is 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.

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 due to 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. Addition of elastomers such as EPDM or SBR improves impact resistance.

For unfilled grades, shrinkage is of the order of .01 - .015 mm/mm (1 - 1.5%). 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.

Major Manufacturers

- BASF (Ultramid B)
- DuPont (Zytel)
- DSM (Akulon)

PA 66 (Polyamide 66, Nylon 66, or Poly(hexamethylene adipamide))

Applications

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

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 750 - 1,250 bar (~11,000 - 18,000 psi), depends on material and product design
Injection Speed:	High (slightly lower for reinforced grades)

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 $0.5 \cdot t$ (where "t" is 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.

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 which 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 filler. 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 - 0.02 mm/mm (1 - 2%). Addition of reinforcing glass fibers reduces the shrinkage to 0.2 - 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.

Major Manufacturers

- BASF (Ultramid A)
- DuPont (Minlon - mineral reinforced grades; Zytel)
- Monsanto (Vydyne).

PBT (Polybutylene Terephthalates)

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)

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 - 8 hours (or 150 C (300 F) for 2 - 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:	225 - 275 C (437 - 527 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 - 120 C / 59 - 248 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 1500 bar / 21750 psi).
Injection Speed:	Fastest possible speeds should be used (due to fast solidification of PBTs)

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 $0.8 - 1.0 * t$ where "t" is the part thickness. When using submarine gates, the minimum recommended diameter is 0.75 mm.

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 and 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 and this could cause warpage due to 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 of the order of 0.015 - 0.028 mm/mm (1.5 - 2.8%). A 30% glass-filled resin has a shrinkage range of 0.3 - 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 - 43 C (71 - 109 F).

The melt viscosity is fairly low and due to fast crystallization rates, cycle times are typically low.

Major Manufacturers

- BASF (Ultradur)
- Hoechst Celanese (Celanex)
- GE Plastics (Lomod, Valox)
- Bayer (Pocan)
- Huls (Vestoduv)
- Mitsubishi Engineering Plastics (Novadur)
- Teijin Chemicals

PC (Polycarbonate)

Typical Applications

- Electronic and business equipment (e.g., computer parts, connectors)
- Appliances (e.g., food processors, refrigerator drawers)
- Transportation (e.g., head lights, tail lights, instrument panels)

Injection Molding Processing Conditions

Drying:	PC resins are hygroscopic and pre-drying is important. Recommended drying conditions are 100 - 120 C (212 - 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
Fill 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

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 (.1 -.2 mm/mm).

High molecular weight PCs (which translate to low melt flow rate) have higher mechanical properties, but processibility 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 processibility, 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 -140 C (266 - 284 F) and the Vicat Softening Point is typically around 155 C (311 F).

Major Manufacturers

- Dow Chemical (Caliber)
- GE Plastics (Lexan)
- Bayer (Apec, Macrolon)
- DSM (Xantar)
- LNP (Lurbricomp, Thermocomp)
- Mitsubishi Engineering Plastics (Novarex)
- Teijin Chemical (Panlite)

PC|ABS

(Polycarbonate-Acrylonitrile-Butadiene-Styrene Blend)

Typical Applications

- Computer and business machine housings
- Electrical applications
- Cellular phones
- Lawn and garden equipment
- Automotive components (instrument panels, interior trim, and wheel covers)

Injection molding processing information

Drying:	Drying is required prior to processing. Moisture content should be less than 0.04 % to ensure stable processing parameters. Drying at 90 - 110 C (194 - 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)
Injection Pressure:	Part dependent
Injection Speed:	As high as possible

Chemical and Physical Properties

PC|ABS offers combined properties of PC and ABS (high processibility 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.

Major Manufacturers

- GE Plastics (Cycloy)
- Dow Chemicals (Pulse)
- Bayer (Bayblend)
- DSM (Stapron C)

- Teijin Chemical (Multilon)

PC|PBT (Polycarbonate | Polybutyleneterephthalate Blend)

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

Injection Molding Processing Conditions

Drying:	110 - 135 C (230 - 275) F for approximately 4 hours
Melt Temperature:	235 - 300 C (455 - 572 F); Depends on specific grade
Mold Temperature:	37 - 93 C (98 - 200 F)

Chemical and Physical Properties

PC|PBT blends offers 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.

Major Manufacturers

- GE Plastics (Xenoy)
- Bayer (Makroblend)

PE-HD (High Density Polyethylene)

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

Injection Molding Processing conditions

Drying:	Not normally necessary if stored properly.
Melt temperature:	220 - 260 C (428 - 500 F); for high molecular weigh resins, the suggested melt temperature range is 200 - 250 C (392 - 482 F)
Mold temperature	50 - 95 C (122 - 194 F) (higher temperatures for wall thickness of up to 6 mm; lower temperature for wall thicknesses greater than 6 mm.)
The cooling rate should be uniform to minimize shrinkage variations. For optimum cycle times, the cooling channel diameters should be at least 8 mm and must be within a distance of 1.3 d from the mold surface (where "d" is the diameter of the cooling channel).	
Injection pressure:	700 - 1,050 bar (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.

Runners and Gates

Diameters of runners range from 4 - 7.5 mm (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.

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 and this is made possible by the use of stereospecific catalysts. Because of molecular regularity, PE-HD has a high level of crystallinity (compared to PE-LD).

Higher levels of crystallinity contribute to higher density, tensile strength, heat distortion temperature, viscosity, and chemical resistance. PE-HD is more resistant to permeability than PE-LD. The impact strength is lower. The properties of PE-HD 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 - 0.925 g/cm³, the material is known as Type 1; Type 2 materials have densities in the range of 0.926 - 0.94 g/cm³, and Type 3 materials have densities in the range of 0.94 - 0.965 g/cm³.

The material flows easily and the MFR ranges from 0.1 - 28. Higher molecular weights (lower MFR grades) have better impact resistance.

Being a semicrystalline material, the molding shrinkage is high (order of 0.015 - 0.04 mm/mm or 1.5 - 4%). This is dependent on the degree of orientation and level of crystallinity in the part (which in turn are dependent 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. PE-HD is soluble in hydrocarbons at temperatures greater than 60 C, but resistance to these materials is greater than that for PE-LD.

Major Manufacturers

- Allied Signal (Paxon)
- BASF (Lupolen)
- Dow (Dowlex)
- DSM (Stamylan HD)
- DuPont (Alathon)
- Exxon Chemical (Escorene)
- Hoechst Celanese (Hostalen)
- Huls (Vestolen)

- ICI (Alkathene)
- Phillips (Marlex)
- Phillips 66 (Marlex)
- Quantum Chemicals
- Union Carbide

PE-LD (Low Density Polyethylene)

Typical Applications

- Closures
- Bowls
- Bins
- Pipe couplings

Injection Molding Processing Conditions

Drying:	Not usually necessary
Melt Temperature:	180 - 280 C (355 - 535 F)
Mold Temperature:	20 - 40 C (68 - 104 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 1,500 bar (21,750 psi)
Pack Pressure:	Up to 750 bar (10,850 psi)
Injection Speed:	Fast speeds are recommended; profiled speeds can limit warpage problems of large surface area parts.

Runners and Gates

All conventional types may be used; PE-LD is well suited for hot runner molds. Insulated hot tip runners are preferred for frequent color changes.

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 - 0.94 g/cm³. PE-LD 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 - 0.05 mm/mm (2 - 5%) when density is between 0.91 - 0.925 g/cm³. When density is between 0.926 - 0.94 g/cm³, the shrinkage is of the order of 1.5 - 4%. Actual shrinkage values are dependent on the molding conditions.

PE-LD is resistant to many solvents at room temperatures but aromatic and chlorinated hydrocarbons cause swelling. Like PE-HD, it is also susceptible to environmental stress cracking.

Major Manufacturers

- Bayer (Baylon)
- BASF (Lupolen/Lucalen)
- Dow (Dowlex LDPE)
- DuPont (Alathon)
- DSM (Stamalyn LD)
- Eastman Chemicals (Tenite PE)
- Exxon (Escorene)
- Novacor (Novapol LD)
- Shell
- Quantum Chemicals

PEI (Polyetherimide)

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, non-implant devices)

Injection molding processing information

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) unreinforced grades 340 - 415 C (644 - 780 F) reinforced grades
Mold Temperature:	107 - 175 C (224 - 347 F); Aim: 140 C
Injection Pressure:	700 - 1500 bar (~10,000 - 22,000 psi) Typical
Injection Speeds:	As high as possible

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 processible. 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.

Major Manufacturers

- GE Plastics (Ultem)

PET (Polyethylene Terephthalate)

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)

Injection molding processing conditions

Drying:	Drying is essential prior to molding. PETs are very sensitive to hydrolysis. Recommended drying conditions are 120 - 165 C (248 - 329 F) for 4 hours. The moisture content should be less than 0.02%.
Melt Temperature:	For unfilled grades: 265 - 280 C (509 - 536 F) For glass reinforced grades: 275 - 290 C (527 - 554 F)
Mold Temperature:	80 - 120 C (176 - 248 F) (preferred range: 100 - 110 C; (212 - 230 F)
Injection Pressure:	(300 - 1,300 bar ; 4,350 - 19,000 psi)
Injection speed:	High speeds without causing embrittlement

Runners and Gates

All conventional types of gates may be used; gates should be 50 - 100% of the part thickness.

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 - 220 C (248 - 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.

Major Manufacturers

- DuPont (Rynite)
- Eastman Chemical (Eastapak)
- Hoechst Celanese (Impet),
- Allied Signal (Petra)

PETG (Glycol-modified PET; Copolyesters)

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

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:	300 - 1,300 bar (4,350 - 19,000 psi)
Injection speed:	High speeds without causing embrittlement

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.

Major Manufacturers

- Eastman Chemicals

PMMA (Polymethyl Methacrylate)

Typical Applications

- Automotive (signal light devices, instrument panels)
- Medical (blood cuvettes)
- Industrial (video discs, lighting diffusers, display shelving)
- Consumer (drinking tumblers, stationery accessories)

Injection Molding Processing Conditions

Drying:	PMMA is hygroscopic and must be dried prior to molding. Drying at 90 C (194 F) for 2-4 hours is recommended.
Melt Temperature:	240 - 270 C (460 - 520 F)
Mold Temperature:	35 - 70 C (90 - 160 F)
Injection Speed:	Moderate

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.

Major Manufacturers

- Cyro Industries

PMMA

- Rohm and Haas (Plexiglass)
- BASF, Mitsubishi Rayon

POM (Polyacetal or Polyoxymethylene)

Applications

Acetals have a low coefficient of friction and good dimensional stability. This makes it ideal for use in gears and bearings. Due to its high temperature resistance, it is used in plumbing (valve and pump housings) and lawn equipment.

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; 190 - 210 C (374 - 410 F) for copolymer
Mold Temperature:	80 - 105 C (176 - 221 F); Higher mold temperatures are preferred for precision molding which reduce post-molding shrinkage
Injection Pressure:	700 - 1,200 bar (~10,000 - 17,500 psi)
Injection Rate:	Medium - High

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.

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 processibility. 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 - 0.035 mm/mm. Differential shrinkage is observed with reinforced grades.

Major manufacturers

- DuPont (Delrin homopolymers)
- BASF (Ultraform copolymers)
- Hoechst Celanese (Hostaform, Celcon copolymers)
- Asahi Chemicals (Tenar)
- PolyPlastics (Duracon)

PP (Polypropylene)

Typical Applications

- Automotive (mostly mineral-filled PP is used: dashboard components, ductwork, fans, and some under-hood components)
- Appliances (doorliners for dishwashers, ductwork for dryers, wash racks and lids for clothes washers, refrigerator liners)
- Consumer products (lawn/garden furniture, components of lawn mowers, sprinklers)

Injection Molding Processing Conditions

Drying:	Not normally necessary if proper storage is used
Melt Temperature:	220 - 275 C (428 - 527 F); not to exceed 275 C
Mold Temperature:	40 - 80 C (104 - 176 F); suggested: 50 C (122 F) The crystallinity level is determined by the mold temperature.
Injection Pressure:	Up to 1,800 bar (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 is highly recommended.

Runners and Gates

In the case of cold runners, typical diameters range from 4 - 7 mm. Full round sprues and runners are recommended. All types of gates can be used. Typical pin gate diameters range from 1 - 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.

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 - 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 - 0.025 mm/mm or 1.8 - 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.

Major Manufacturers

- Amoco Chemicals
- A. Schulman (PolyFort)
- Asahi Chemicals (Loymer, PolyPro)
- BASF (Novolen)
- Borealis
- DSM (Stamylan P)
- Eastman (Tenite)
- Exxon
- Himont (Profax)
- Hoechst Celanese (Hostalen PP)
- Huls (Vestolen)
- ICI (Propathene)
- Idemitsu
- Mitsubishi Chemicals
- Phillips Petroleum (Marlex)
- Shell
- Solvay (Eltex P)

PPE|PPO (Polypropylene Ether Blends)

Typical Applications

- Household appliances (dishwasher, washing machine)
- Electrical applications, such as control housings, fiber-optic connectors

Injection Molding Processing Conditions

Drying:	Recommend drying before molding for approximately 2 - 4 hours at 100 C (212 F). PPOs have low levels of moisture absorption 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:	600 - 1,500 bar (8,700 - 21,750 psi)

Runners and Gates

All gates can be used; tab and fan gates are preferred

Chemical and Physical Properties

PPO is poly(2,6 dimethyl p-phenylene) oxide. The ether linkages offer easier processibility. 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 processibility 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.

Major Manufacturers

- GE Plastics (Noryl)

PPE|PPO

- BASF (Luranyl)
- Asahi Chemicals (Xyron)
- Huls (Vestoran)

PS (Polystyrene)

Typical Applications

- Packaging
- Housewares (tableware, trays)
- Electrical (transparent housings, light diffusers, insulating film)

Injection Molding Processing Conditions

Drying:	Not usually required unless stored improperly. If drying is needed, the recommended conditions are 2-3 hours at 80 C (176 F)
Melt Temperature:	180 - 280 C (356 - 536 F); upper limit is 250 C for flame retardant grades (19 - 158 F)
Mold Temperature:	Suggested: 40 - 50 C (104 - 122 F)
Injection Pressure:	200 - 600 bar (3,000 - 8,700 psi)
Injection Speed:	Fast speeds are recommended

Runners and Gates

All types of conventional gates may be used.

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 - 0.7%.

Major Manufacturers

- Asahi Chemicals
- BASF (Polystyron)
- Chevron
- Chi Mei Industrial (Polyrex)

- Dow (Styron)
- Elf Atochem
- Fina Oil and Chemicals
- Huls (Vestyron)
- Huntsman Chemical Corporation
- Idemitsu (Styrol)
- Mitsubishi Chemicals.
- Mobil

PVC (Polyvinyl Chloride)

Typical Applications

- Water distribution piping
- Home plumbing
- House siding
- Business machine housings
- Electronics packaging
- Medical apparatus
- Packaging for foodstuffs

Injection Molding Processing Conditions

Drying:	Not usually necessary as PVC absorbs very little water.
Melt Temperature:	185 - 205 C (365 - 400 F)
Mold Temperature:	30 - 50 C (86 - 122 F)
Injection Pressure:	Up to 1,500 bar (21,750 psi)
Packing Pressure:	Up to 1,000 bar (14,500 psi)
Injection Speed:	Relatively slow, to avoid material degradation

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 - 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.

Chemical and Physical Properties

Rigid (unplasticised) 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 processible. 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, unplasticised 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 - 0.006 mm/mm or 0.2 - 0.6%).

Major Manufacturers

- BASF (Vinidur, Vinoflex)
- BF Goodrich
- Formosa Plastics
- Geon
- Georgia Gulf
- Huls (Vestolit)
- Neste
- Schulman (Polyvin)
- Shell (Rovin)
- Vista Chemicals

SAN (Styrene Acrylonitrile)

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

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 2-4 hours prior to molding.
Melt Temperature:	200 - 270 C (392 - 518 F); 230 - 260 C (446 - 500 F) for most applications; lower end of the range is used for molding thick wall 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 (140F). The cooling system must be well designed because the mold temperature affects the parts appearance and shrinkage and warpage.
Injection Pressure:	350-1,300 bar (5,000 - 20,000 psi)
Injection Speed:	High speeds are recommended

Gates

All conventional gate types may be used. The gates must be of proper size which aid in processing and do not cause streaks, burn marks, or voids.

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 processibility and the acrylonitrile component imparts chemical and thermal resistance.

They have excellent load bearing capacity, rigidity, good resistance to chemicals, heat deformation, and cyclic temperature loads, and dimensional stability. The properties are dependent 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 - 0.007 mm/mm (0.3 - 0.7%).

Major Manufacturers

- BASF (Luran)
- Dow Chemical (Tyril)