

10 Shrinkage and Warpage

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

10.1 Injection Molding and Shrinkage

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

10.1.1 What Are Shrinkage and Warpage?

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

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

10.1.2 Shrinkage and Machine Settings

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

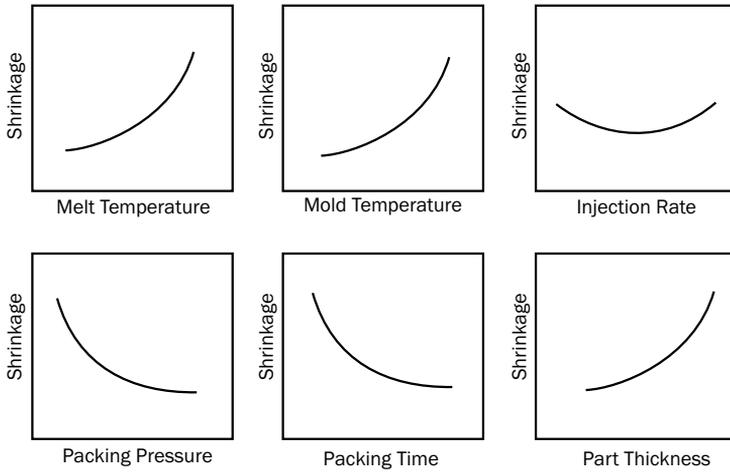


Figure 10.1 Effect of machine settings on shrinkage

10.1.3 Mold Filling and Packing

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

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

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

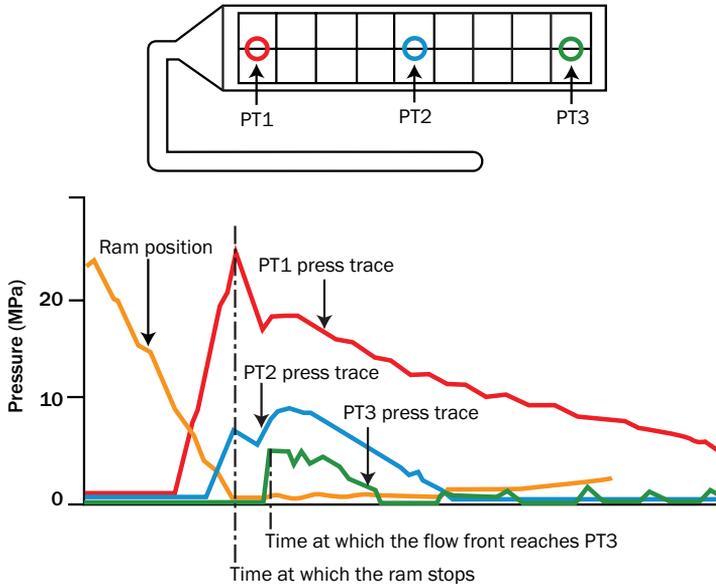


Figure 10.2 Pressure traces for a simple molding

10.1.4 How Pressure and Time Affect Shrinkage

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

10.1.4.1 Shrinkage near the Gate

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

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

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

10.1.4.2 Shrinkage at the End of Flow

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

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

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

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

10.1.5 Thermally Unstable Flow

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

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

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

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

10.2 Basic Causes of Shrinkage and Warpage

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

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

10.2.1 Causes of Shrinkage

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

10.2.1.1 Volumetric Shrinkage

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

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

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

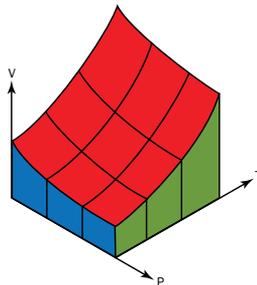


Figure 10.3 3D PVT diagram

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

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

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

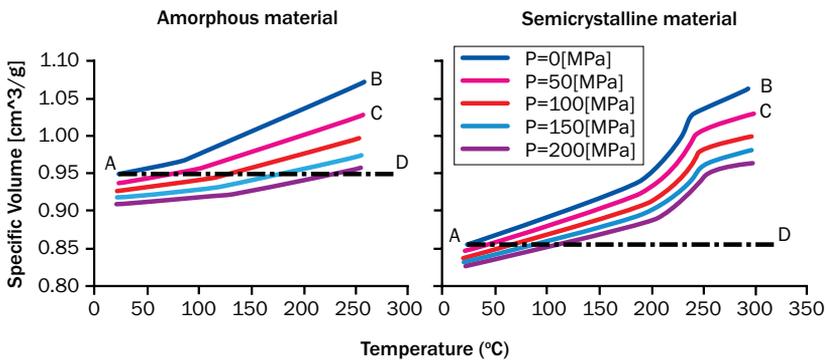


Figure 10.4 PVT diagrams for polymers

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

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

10.2.1.2 Relationship between Linear and Volumetric Shrinkages

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

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

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

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

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

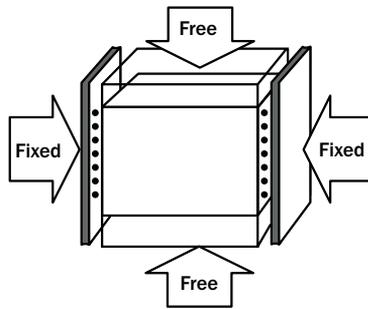


Figure 10.5 Effect of mold restraint

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

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

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

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

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

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

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

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

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

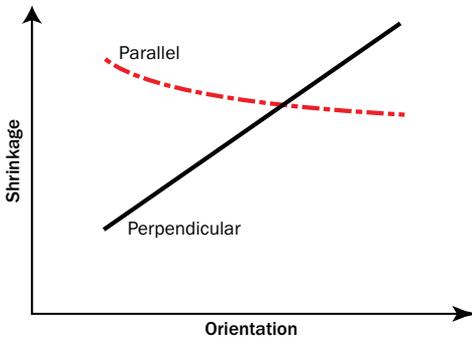


Figure 10.6 Parallel and perpendicular shrinkage of polypropylene

10.2.1.3 Summary

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

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

10.2.2 Causes of Warpage

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

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

- Orientation effects
- Area shrinkage effects
- Differential cooling effects

10.2.2.1 Orientation Effects

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

10.2.2.2 Area Shrinkage

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

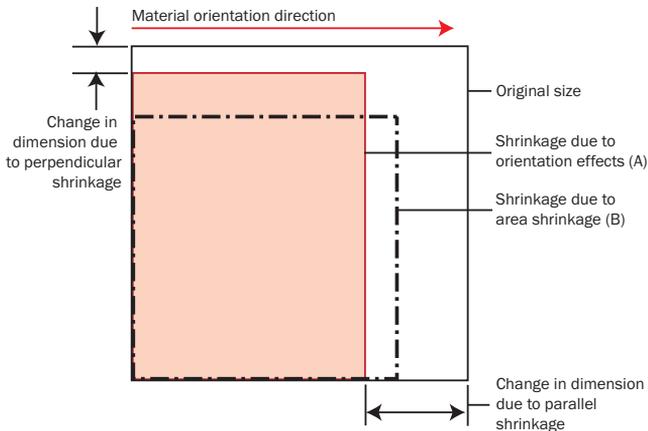


Figure 10.7 Definition of area shrinkage

10.2.2.3 Differential Cooling

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

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

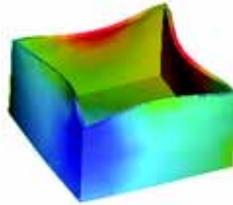


Figure 10.8 Warpage caused by differential cooling

10.2.3 Relating Orientation and Area Shrinkage to Warpage

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

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

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

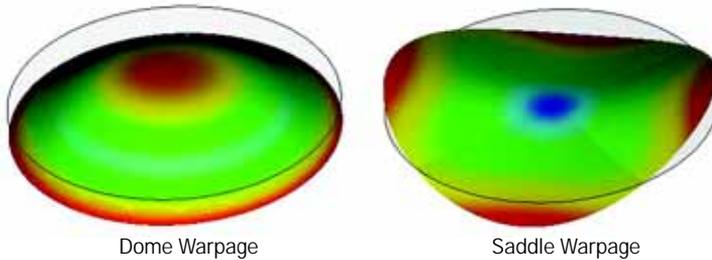


Figure 10.9 Dome- and saddle-type warpage for a disk

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

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

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

10.3 Designing Accurate Parts Considering Warpage

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

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

In this section we discuss each of these areas.

10.3.1 Material Selection

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

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

10.3.2 Wall Thickness Variation

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

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

10.3.2.1 Crystalline Materials

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

10.3.2.2 Amorphous Materials

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

10.3.2.3 Using Wall Thickness to Control Shrinkage near Gates

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

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

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

10.3.2.4 Using Wall Thickness to Control Orientation and Area Shrinkage

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

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

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

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

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

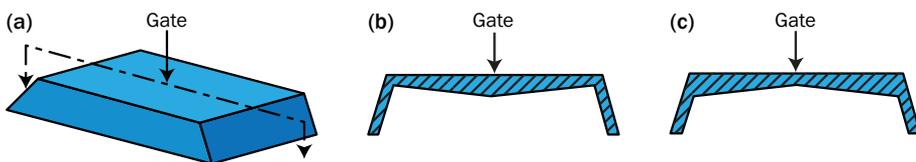


Figure 10.10 Using wall thickness to control warpage

10.3.3 Gate Position and Runner Dimensions

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

The idea is to first try to position a single gate in the flow centroid of the molding (Figure 10.11 (a)). If a single gate is not practical, then the molding is mentally broken up into

sub-moldings, gates positioned at the flow centroid of each sub-molding, and the runner system then dimensioned such that each sub-molding is filled at the same instant in time, as shown in Figure 10.11(b). The essence of this system is to avoid problems associated with overpacking, such as variation in shrinkage, part sticking in the cavity, etc.

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

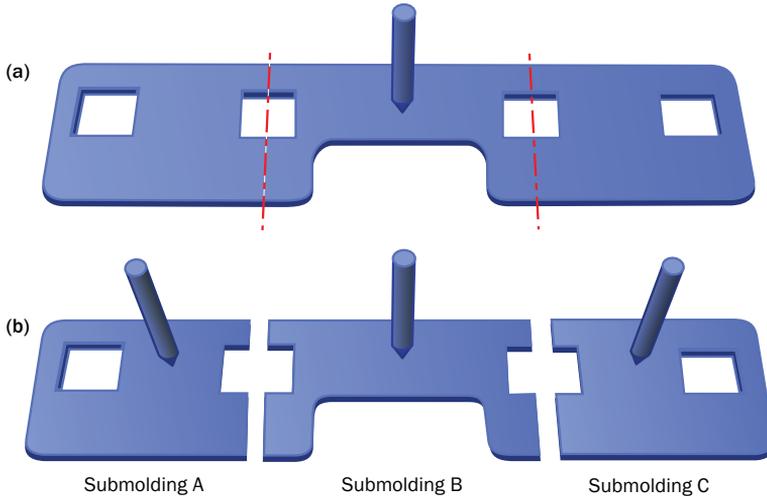


Figure 10.11 Gating and flow centroids

10.3.3.1 Design Criteria for Number and Position of Gates

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

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

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

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

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

10.3.4 Molding Conditions

The basic principles of establishing molding conditions as developed in Chapter 2 remain valid. However, with the common use of injection profiles there is a need to revise these principles to take advantage of them.

Traditional design principles aim to have a uniform temperature over the cavity, that is, frictional heat is balanced against heat loss by conduction. In practice, most heat is generated around the gate areas, so the temperatures rise around the gate to a maximum and then drop off toward the end of flow. So the criterion is that the temperature at the end of flow should equal the temperature at the gate.

The aim in designing runner systems is to run the barrel temperature low to minimize degradation, and then use the runner system to generate frictional heat. This criterion does not consider problems such as jetting and other surface defects which tend to occur around the gate area. With programmed injection, the aims are to fill the cavity in the shortest possible time and to give a high temperature at the end of flow without causing jetting, overheating, or similar problems.

In addition to these molding constraints, there are also mechanical constraints arising from the flow rate and clamp tonnage capabilities of the machine. The ideal profile is generated by using the maximum injection rate and clamp tonnage of the machine as upper bounds, and then reducing the flow rate locally so that material constraints, such as maximum temperature, shear rate, and shear stress, are not exceeded.

It should be noted that the aim is to fill the mold in the shortest practical time and that by deliberately slowing the injection rate in certain key areas, normally around the gate and at the end of flow, a much shorter injection time can be achieved. In many cases, the available clamp tonnage will require a slower injection rate at the end of fill.

In the packing stage the aim is to bring the extremities of the cavity up to pressure without exceeding clamp tonnage limitations. A well-designed feed system should allow the cavity to be pressurized rapidly. The constant clamp tonnage criterion will usually result in the packing pressure decaying with time. This clamp tonnage is held constant until the plastic at the end of flow has frozen or until the required volumetric shrinkage has been achieved. In reality, these two criteria give the same results. The freeze front should then gradually progress back toward the gate. At each point in time, the packing pressure can then be adjusted to give the required volumetric shrinkage throughout the part.

10.3.4.1 Design Criteria for Volumetric Shrinkages

Volumetric shrinkages of molded material can vary widely. It is possible to produce negative volumetric shrinkages. In practice this means the part will actually expand on removal from

the mold. This effect occurs if there is residual pressure inside the part at the end of the molding cycle. Typically negative volumetric shrinkages are associated with excessive packing pressure. At the other extreme, volumetric shrinkages can be over 15%.

Molding conditions that produce low or negative volumetric shrinkages are likely to be associated with high levels of orientation and hence high residual stress levels.

Very high volumetric shrinkages will usually result in excessive shrinkages in the thickness direction, which may be seen as poor surface finish or sink marks, even though shrinkages in the plane of the molding may still be within the normal range.

Under these conditions, mold constraint will be setting up very high stress levels in the part as the plastic is stretched and thinned within the cavity. Either of these extremes should be avoided.

10.3.4.2 Uniform and Acceptable Volumetric Shrinkage

The first step in the shrinkage design approach is to try to achieve a uniform and acceptable volumetric shrinkage throughout the part. Simply ensuring that the part will fill does not automatically guarantee that the shrinkages will be acceptable, particularly at the end of flow.

In Chapter 2, it was recommended that both pressure and temperature be limited. The pressure required to fill the cavity should be significantly less than the pressure available on the machine. The temperature at the end of flow should be adequately high, preferably close to the melt temperature at the entrance of the cavity. Experience has shown that these design principles have been reliable, if perhaps a little conservative. The aims can now be redefined in terms of volumetric shrinkage. The new criteria are to achieve both uniform and acceptable values of volumetric shrinkage over the cavity. Design values for acceptable volumetric shrinkage are determined as part of the material testing procedure.

10.3.4.3 Minimizing Sink Marks

These ideal conditions cannot be obtained in some parts, particularly those with thick and thin sections or very long flow paths. In general, thick sections should not be filled through thin sections, but if this is impossible to avoid, such as with bosses, an alternative molding strategy must be adopted.

Instead of starting off with the highest pressure available within clamp tonnage limitations, the pressure at the end of fill should be lowered so there is no backflow out of the cavity. The pressure is then held at a low value to allow the frozen layer to grow and then the pressure gradually raised to maintain a constant flow rate, which preserves a uniform frozen layer thickness. Pressure is raised until the maximum clamp tonnage criterion is reached and then the pressure decayed again using the same criteria of maintaining a constant volumetric shrinkage.

 This procedure, while effective for eliminating sink marks and increasing pack at the end of flow, gives rise to moldings with significant orientation and hence stress and should, therefore, only be used when there is no other alternative.

10.3.4.4 Melt Temperature

Increasing melt temperature may increase or decrease shrinkage. Typically, very low melt temperatures lead to relatively high shrinkage because the part cannot be adequately packed out. Increasing the temperature, up to a certain point, decreases the shrinkage because the viscosity is reduced and so packing pressure can be better distributed through the cavity. For a given packing pressure and time, further increases in melt temperature lead to higher shrinkages as the cavity is then filled with material of relatively low density.

10.3.4.5 The Effect of Molding Conditions on Orientation

The above criteria will minimize variation in volumetric shrinkage throughout the part, but does not directly consider the effect of orientation. Orientation is caused by the combination of shearing while freezing. In general, the conditions of filling the cavity as rapidly as possible within constraints will reduce orientation.

The majority of the orientation occurs in the packing phase, so if orientation is diagnosed as the major cause of warpage, then high melt temperatures (probably in conjunction with faster injection), coupled with higher but still uniform volumetric shrinkage, should be used.

10.3.4.6 Mold Temperature

Variations in mold temperature are a well-known cause of warpage. The classic bowing in of the sides of boxes is normally due to differences in temperature between the core and cavity.

If a part is warping due to poor cooling, the solution method is usually obvious, that is, repositioning of coolant lines, use of beryllium copper inserts, or heat pipes, increasing coolant flow, etc.

The aim when designing a cooling circuit is not necessarily to achieve a uniform mold temperature. The aim should be to achieve a uniform cooling time. This means that wherever practical, thicker areas of the part should have a lower mold temperature, while thinner areas should be deliberately run at a higher mold temperature, possibly by moving the cooling lines further away from the surface. A higher mold temperature will give more uniform shrinkage and lower orientation levels, but will require much longer cooling times. Raising mold temperature without increasing cooling times results in worse moldings because of the lower mold constraint time.

10.3.4.7 Packing Pressure

A packing pressure that is initially high and then reduces with time gives more uniform area shrinkage and decreases orientation. Generally such profiles are optimum for parts with little variation in wall thickness.

In the case of a thin section feeding a thicker region, the above approach will not be optimum. A high degree of pack can be obtained using a packing pressure that is initially low but increases with time. This minimizes sink marks in the thick region but causes high orientation. Often a better solution is obtained by a change in gating position.

10.3.5 Cooling Line Layout

It is a mistake to design cooling circuits based on getting cooling lines as close to the part wherever possible. Variations in mold temperature can be a very useful method for compensating for other sources of warpage. For example, centrally gated parts often have a tendency to twist or saddle warp due to a combination of high orientation at the gate and low shrinkages from overpacking. This can be offset by deliberately running the extremities of the mold very hot (to increase shrinkage), while running the gate area cold (to reduce shrinkage).

It should be noted that the heat load around the gate is much higher due to the longer flow time of the plastic and relatively high frictional heating. To achieve a temperature distribution in which the outside is hot and the gate region is cold may require a much distorted cooling circuit or, more realistically, twin circuits with their own temperature control.

Temperature differential on either side of the mold can be deliberately used to make a part deflect in a certain direction. A common example is a part with ribs that are thinner than the main surface. There is a natural tendency for the part to deflect away from the thinner ribs. By running the rib area hotter, the part can be deflected back to the required shape. In some cases, an insert of lower conductivity or containing a separate cooling circuit, can be used to form the ribs. This gives better temperature control of the rib temperature.

11 Moldflow Design Procedure

- Determine analysis objectives
- Moldflow analysis steps framework
- Using Moldflow to evaluate an initial design
- Using Moldflow to optimize the design

11.1 Determine Analysis Objectives

Every analyzed part has a different set of constraints in the form of objectives, restrictions, and guidelines. These constraints must be taken into consideration when doing an analysis. However, you do not want to over-restrict the analysis to the point where a solution cannot be found. Moldflow can help identify problems and solutions to those problems, but there must be flexibility in the design to allow for solutions not initially considered.

The objectives defined for a part are as varied as the parts that can be injection molded. However, below is a list of analysis objectives that you might have. Some are just flow analysis related; others will require cooling and warpage analysis.

- Will the part fill?
- What material will work best for my part with regard to fill properties (e.g. pressure, shear stress, and temperature distribution)?
- What processing conditions should be used to mold this part?
- Where should the gate be located?
- How many gates are required?
- Where will the weld lines be, and will they be of high quality?
- Will there be any air traps?
- How thick can the part be made?
- Is the flow balanced within the part with the fixed gate location?
- Are the ribs too thin to fill completely?
- Are the ribs so thick that they shrink too much?
- Can the part be packed out well enough?
- Will this snap fit break during use?

- Can the part be filled and packed in the press specified for the job?
- Are the runners balanced?
- What size do the runners need to be to balance the fill?
- Is the runner volume as small as it can be?
- Is the gate too big or too small?
- Can a 10-second cycle time be achieved?
- Will the part warp more than the 1.5 mm tolerance?
- Will the parts molded in a family tool assemble together?

 This is not a comprehensive list, but it gives you an idea of what can be done.

11.2 Moldflow Analysis Steps Framework

The exact steps taken to conduct an analysis on a part are as varied as the parts and problems to be solved. Results from an analysis will guide you to a solution path that will change depending on the problem and choices you make to fix the problem. Generally, there are several ways to solve a problem, some better than others. Following are the basic steps for conducting flow, cooling, and warpage analyses on a part.

11.2.1 The Whole Process

The procedure for analyzing a part from filling to warpage is charted in Figure 11.1. It follows the Moldflow design principles discussed in Chapter 4: the filling of the part, followed by the runners, cooling system, packing, then warpage. Each of these basic steps will be discussed in detail in the following sections.

11.2.2 Optimize Fill

Optimizing the filling of the part is the first major step in part optimization. This is the foundation for all other analysis work. The step therefore includes determining the objects as described above and preparing the model. The steps to optimize the fill are shown in Figure 11.2 and then described.

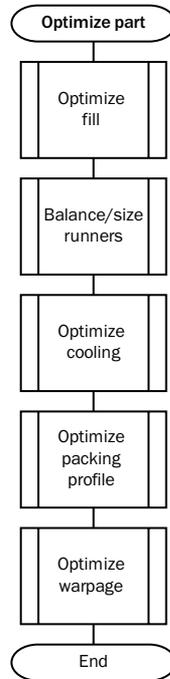


Figure 11.1 The basic steps for optimizing the part from fill to warpage

11.2.2.1 Prepare the Model

Preparing the model involves importing from a CAD system the geometry of the part and meshing it inside Moldflow, or importing a finite element mesh directly. This is discussed in Chapter 5.

11.2.2.2 Select the Material

To run an analysis in Moldflow, there are many material properties that have to be specifically tested for. What needs testing depends on the analysis to be performed, but generally information about the material is needed in the following categories:

- Rheological properties
- PVT properties
- Thermal properties
- Filler properties
- Mechanical properties
- Shrinkage properties
- Recommended processing conditions

Moldflow has a very extensive database with the required properties needed for analysis. If a material is not found, the data can be obtained from Moldflow’s material testing services, the material manufacturer, or other sources.

 The material properties are a key input into the analysis. If poor material data is used, the results from the analysis will not be as reliable.

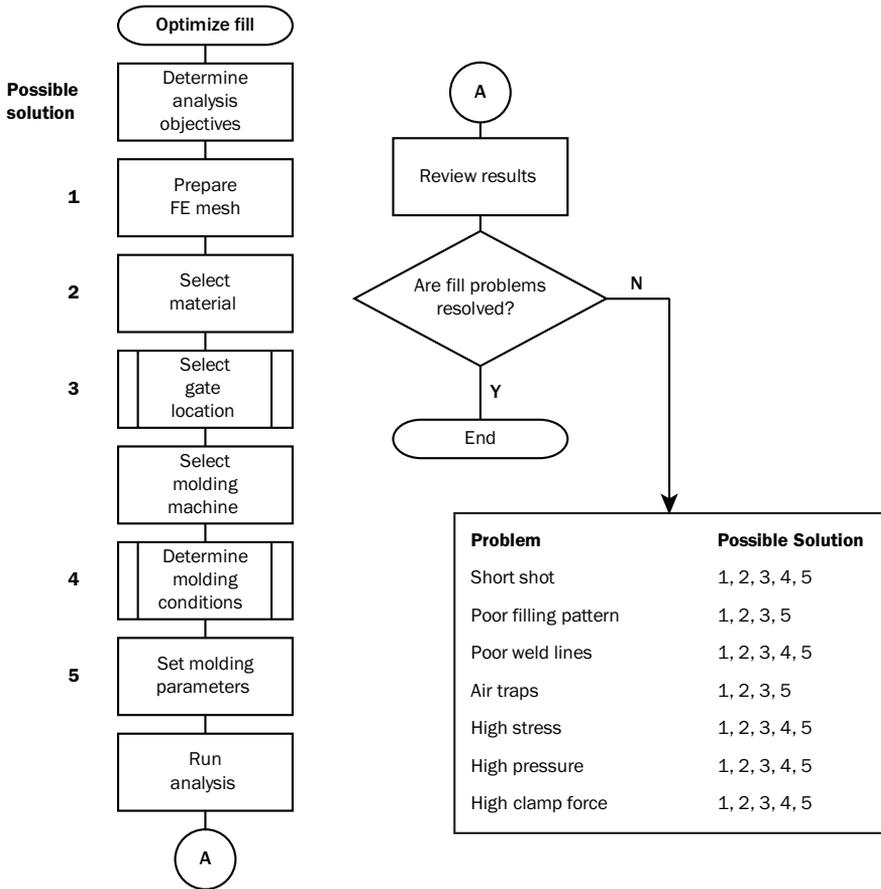


Figure 11.2 The steps required to optimize the filling of the part

11.2.2.3 Select Gate Location(s)

The number and location of the gates need to be determined. Many times this is not known and is one of the reasons Moldflow is being used. Some initial gate location must be defined. Figure 11.3 outlines the procedure. Additional information on gate locations is discussed in Chapter 7.

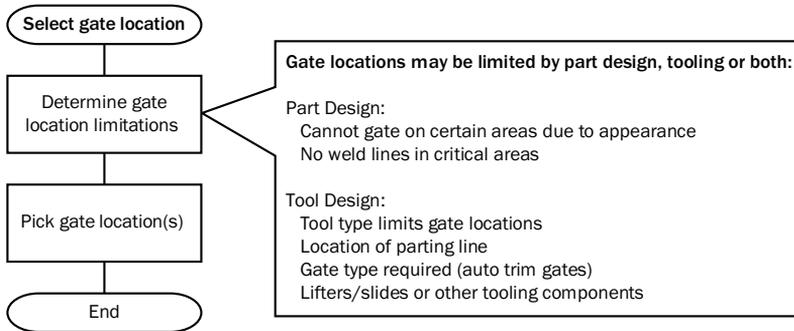


Figure 11.3 Selecting a gate

11.2.2.4 Select a Molding Machine

To run an analysis, some information about the molding machine being used must be defined: at least the injection pressure capacity and clamp tonnage capacity. Every analysis has a default molding machine defined that can be used in most cases. Moldflow also has an extensive database of molding machines so the specific machine can be used. For both the default and specific molding machines, all the parameters of that machine can be customized.

11.2.2.5 Determine the Molding Conditions

The molding conditions including the following must be defined:

- Mold temperature
- Melt temperature
- Injection time

These are the fundamental molding conditions needed for a fill or flow analysis. Optimizing these conditions is often a critical step for the part analysis and is outlined in Figure 11.4. Optimizing molding conditions often involves looking at the gate locations also. Discussion of molding conditions is found in Chapter 2.

11.2.2.6 Set Molding Parameters

Molding parameters include other inputs needed for the basic filling analysis. Many times default values are used, at least initially. Key parameters include the method and timing of the velocity to pressure switchover and the packing profile.

11.2.2.7 Run the Analysis

The analysis in this case is a filling analysis that only looks at the filling phase of the cycle. The analysis ends when the part is 100% full. The filling phase is the phase of the cycle where most problems will occur and/or be fixed.

Initial work on the part does not include the runner system, it only involves the part. This is discussed in Chapter 4.

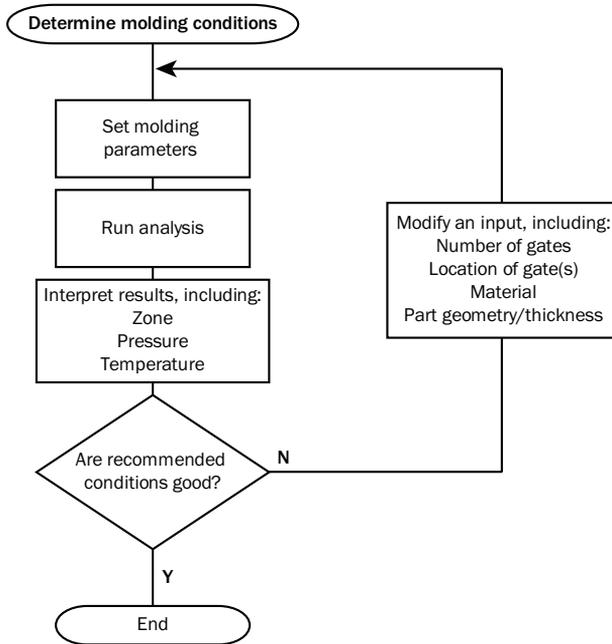


Figure 11.4 Determine molding conditions

11.2.2.8 Review the Results

When the filling analysis is done, the results of this analysis are reviewed to find any problems and to help determine how to proceed. There are a significant number of results that can be reviewed, but most are not used for every analysis. Which results are reviewed will depend on what the analysis objectives are and what problems are found.

11.2.2.9 Resolve Any Filling Problems

The results will highlight many problems. Figure 11.2 has a section listing just a small number of the potential problems a filling analysis can be used to find. The possible solution column lists different steps that may resolve the problem.

Resolving a filling problem is an iterative process. Never will just one fill analysis be enough. There will always be some question that needs to be answered requiring additional analysis. Depending on the nature of the problem and the proposed solutions, only two or three analyses may need to be run; in other cases many more iterations may be needed.

11.2.3 Balance and Size the Runners

After the filling of the part has been optimized, the sizing and balancing of the runner system needs to be done. Depending on the layout and complexity of the runner system, the runners simply may need to be sized to minimize volume, and other times the runners must be balanced to achieve the required filling pattern within and between parts. Chapter 8 discusses runner design and specifically sizing and balancing. The steps for balancing runners are shown in Figure 11.5.

11.2.3.1 Add Runners to the Part

Runners are added to the model once the part has been optimized. In most cases, the runner system is modeled within Moldflow and not imported.

 In some cases when the part has multiple gates and the part is not symmetric, the sizing of the runner system is integral to the part optimization. In these cases, runner systems are added to the part model before the part is completely optimized.

11.2.3.2 Determine Optimized Molding Conditions

During the part optimization process, the molding conditions for the part are optimized. Now that a runner system is being added, these conditions need to be modified to take into account the volume of the runners and the shear heat developed in the runners.

The volume of the runner needs to be accounted for because of the time required to fill the runners. Most of the time, the filling of the part is done by specifying an injection time. This time must be converted to flow rate by the following:

$$\text{Flow rate} = \frac{\text{Volume} \times \text{No. of Cavities}}{\text{Injection time}} \quad (11.1)$$

The volume is the volume of one cavity.

By using flow rate, whatever the volume of the runners will be, the fill time for the part itself will always be the optimum value. The total fill time will go up because of the volume of the runners.

The shear heat generated in the runners will change as the size of the runners change. However, the temperature entering the part should be within 5°C of the optimized melt temperature determined in the filling analysis for the part. The shear heat in most runner systems will be more than 5°C. It could be 30°C or more. Run a filling analysis to determine the amount of shear heat. The temperature entering the sprue is then lowered by the amount of shear heat. This becomes a good starting point for further runner optimization.

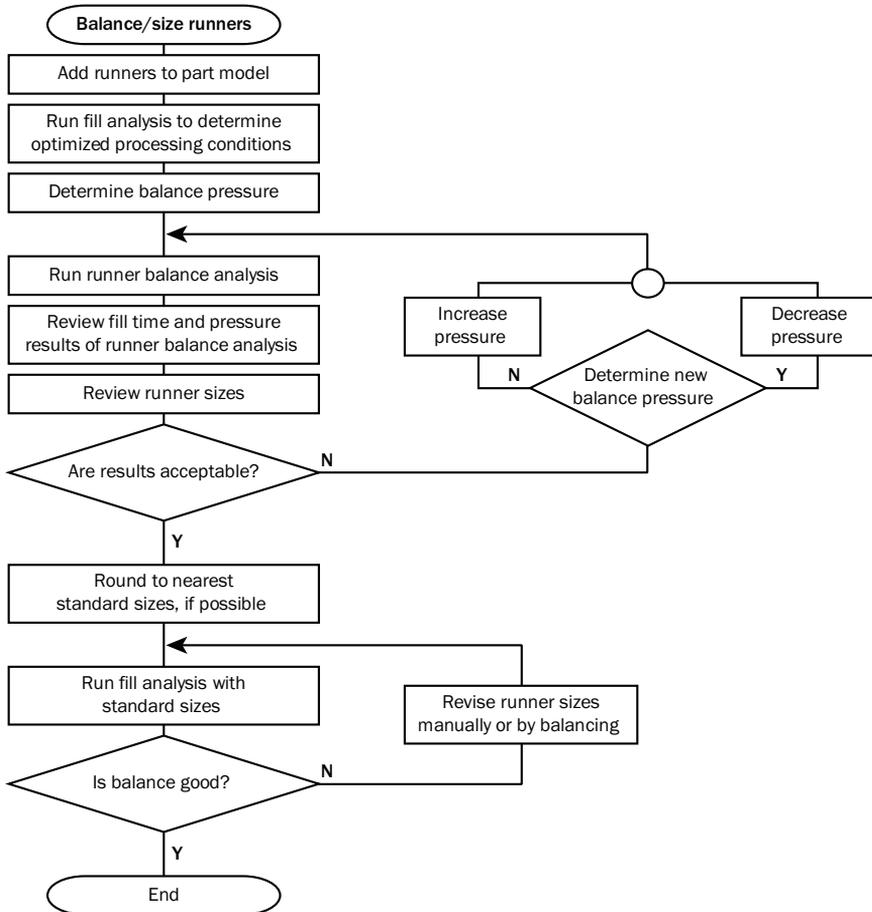


Figure 11.5 Balance and size the runners

11.2.3.3 Determine Balance Pressure

When the automatic runner balancing is done, a key input is the balance pressure. The analysis will change the size of runners so the pressure to fill the mold (runners and parts) is within the tolerance from the balance pressure. Even for geometrically balanced runner systems, the runner balance analysis can be used to downsize the runners to save volume and so the mold takes an acceptable amount of pressure.

When the runners are artificially balanced, the balance pressure becomes a critical input. If the pressure is too high, the runners will become too small, if the pressure is too low, the runners will become too large.

11.2.3.4 Review Runner Balance Results

Once the runner balance analysis is done, the fill analysis results can be reviewed. The balance is mostly determined by looking at the filling pattern and pressure results. If the results are not satisfactory, the balance analysis is done again with a revised balance pressure.

11.2.3.5 Review Runner Sizes

If the balance is acceptable, the sizes of the runners are viewed. The balance may be good, but the runner sizes may be too large or small. This would require another balance analysis with a new balance pressure.

11.2.3.6 Round Runner Sizes

Once the runner balance is acceptable, the sizes of the runners can be compared to standard sizes. The runner balance does not consider the size of the runner in relation to standards: it wants to make sure the parts fill uniformly.

If the sizes of the runners determined by the balance analysis are close to standard sizes, the runners can be changed to a standard size. The more the runners are changed the more “out of balance” the mold will become.

11.2.3.7 Run Analysis with Standard Sizes

If the runner sizes were manually changed to a standard size, a filling analysis should be run to ensure the filling of the part is still acceptable.

11.2.4 Optimize Cooling

Once the runner sizing/balancing is done, the cooling of the mold can be considered. Figure 11.6 shows the steps required to optimize cooling.

11.2.4.1 Determine Analysis Objectives

The cooling analysis objectives are the most important part of the process. What problems are to be investigated and/or solved? Generally, the mold surface temperature should be made as uniform as possible and the cycle time should be minimized.

11.2.4.2 Model Cooling Components

To run a cooling analysis, the cooling channels must be modeled. Possibly inserts can be modeled if high conductivity inserts are used. Other components can be modeled as well. Chapter 9 discusses design rules for placing cooling channels. Cooling components are normally modeled in Moldflow, but they can be imported as well.

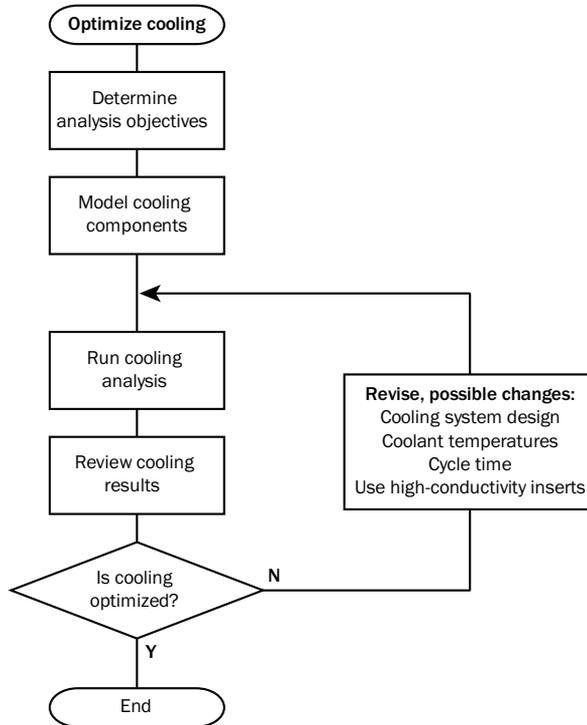


Figure 11.6 Optimize cooling

11.2.4.3 Review Cooling Results

The mold surface temperature is the most important result from cooling—the more uniform the results, the better. Other results from a cooling analysis help determine why there are problems with the temperature distribution.

11.2.4.4 Revise Inputs to the Cooling Analysis

When the cooling results are not acceptable, something needs to change. Generally, a cooling analysis is run to help place the coolant lines correctly and to determine coolant temperatures, flow rates, etc. These parameters are often revised in the process of optimizing the cooling.

11.2.5 Optimize the Packing Profile

Packing is best done after a cooling analysis because the packing and compensation phases are heat transfer dominated. When a packing analysis is conducted after cooling analysis, the tool's ability to extract heat from the part is accurately modeled. Figure 11.7 shows the steps involved in optimizing a packing profile.

The primary output from a packing analysis is the volumetric shrinkage. The amount and distribution of the volumetric shrinkage is critical for determining the linear shrinkage and warpage of the part. To optimize the volumetric shrinkage, the range of volumetric shrinkage is minimized.

11.2.5.1 Determine Initial Packing Pressure and Time

The maximum packing pressure that can be used is related to the machines clamp force and is determined by

$$P_{max} = \frac{\text{Machine Clamp force limit}}{\text{Total projected area of the shot}} \times \text{Unit conversion} \times 0.8 \tag{11.2}$$

where:

- P_{max} = The maximum packing pressure that could be used.
- Machine clamp force limit = Tonnes(metric) or tons(US / English units)
- Total projected area of the shot = cm² or inches²
- Unit conversion = 100 for metric units, 2000 for english units
- 0.8 = Safety factor to only use 80% of machine capacity

Often the maximum pressure that could be used based on the equation above is much higher than is needed. Packing pressurizing are generally 80 to 100% of the pressure required to fill the part. Packing pressures, however, can be much higher or lower however.

An initial estimation of packing pressure is often changed in the process of optimization.

11.2.5.2 Review Results

Volumetric shrinkage, pressure traces, and gate freeze time are the results most often reviewed to determine whether the shrinkage is acceptable or not. If the shrinkage is not acceptable, these results are used to determine how the packing profile must be changed to improve the volumetric shrinkage. To fully optimize the volumetric shrinkage, the process is iterative.

11.2.6 Optimize Warpage

Optimizing warpage, shown in Figure 11.8, is the last major step in part optimization. It encompasses all the steps previously discussed. The decisions made previously will influence the results here. In the process of reducing warpage, the solution may require going back to any of the previous steps defined earlier.

However, when design guidelines are properly followed, warpage will be minimized, and the process of warpage optimization becomes a validation process that previous work was well done.

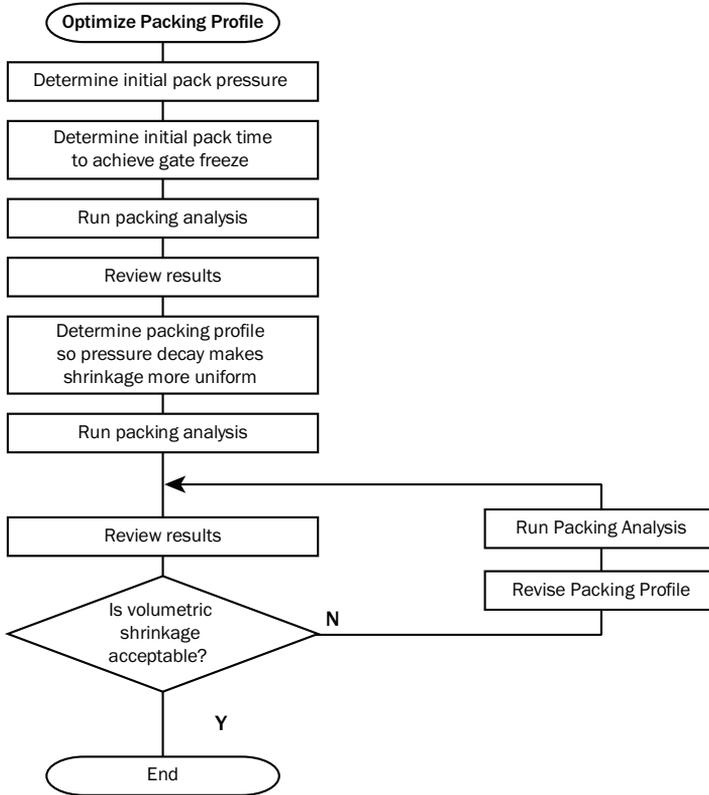


Figure 11.7 Optimizing the packing profile

11.2.6.1 How Warpage Is Defined

One of the most significant challenges when doing a warpage analysis is defining how warpage is measured on the molded part. This needs to be well understood so that results from the warpage analysis can be compared to how molded parts will be measured. Without this understanding, the warpage analysis will be improperly utilized.

11.2.6.2 Determine the Magnitude of Warpage

The first step is measuring how much the part has warped with the current processing conditions. The results from the warpage analysis are compared to the tolerances for the part to determine if the part is acceptable or not. When the parts don't meet the tolerances, the warpage must be reduced.

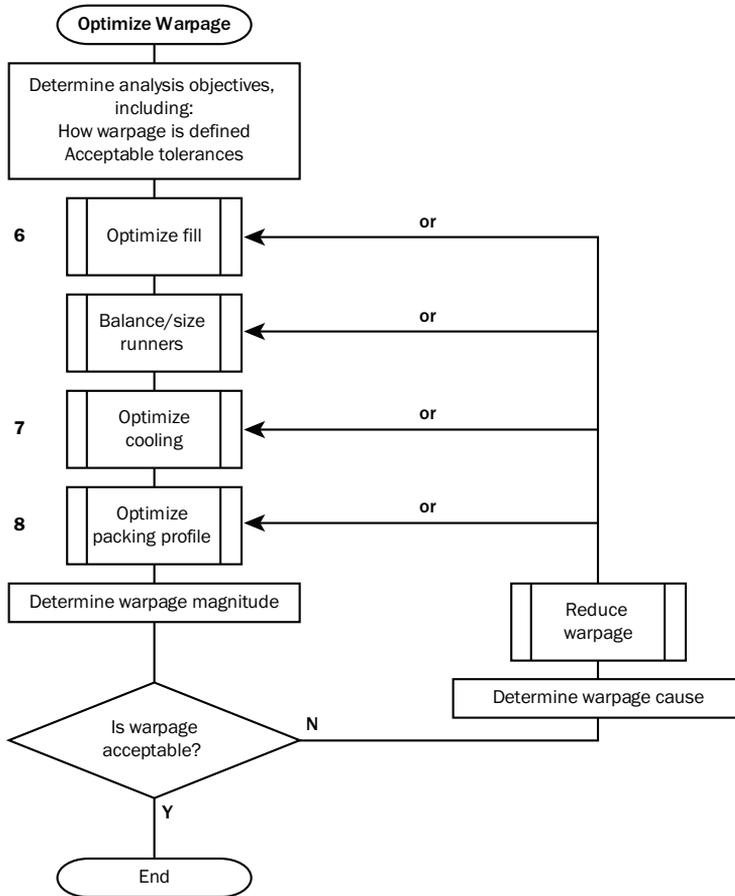


Figure 11.8 Optimize warpage

11.2.6.3 Determine the Cause of Warpage

If the part warps more than the tolerance allows, the warpage must be reduced. Chapter 10 discusses the complex interaction of part geometry, material, and processing parameters that lead to warpage. However, warpage can be broken down to three main causes as discussed in Section 10.2.2, including:

- Orientation effects
- Area shrinkage effects
- Differential cooling effects

Once these three main causes of the warpage can be determined, the warpage can be reduced.

11.2.6.4 Reduce Warpage

Reducing the amount of warpage to get it below the tolerance involves addressing the primary cause of warpage as shown in Figure 11.8. Fixing the primary cause of warpage can be done in many ways. The numbers beside possible ways to fix the warpage refer to steps numbered in Figure 11.9.

Often, reducing the amount of warpage is an iterative process. Several possible solutions could be found and evaluated to determine which one is most practical or economical. Many times it will take several iterations before the warpage is reduced enough.

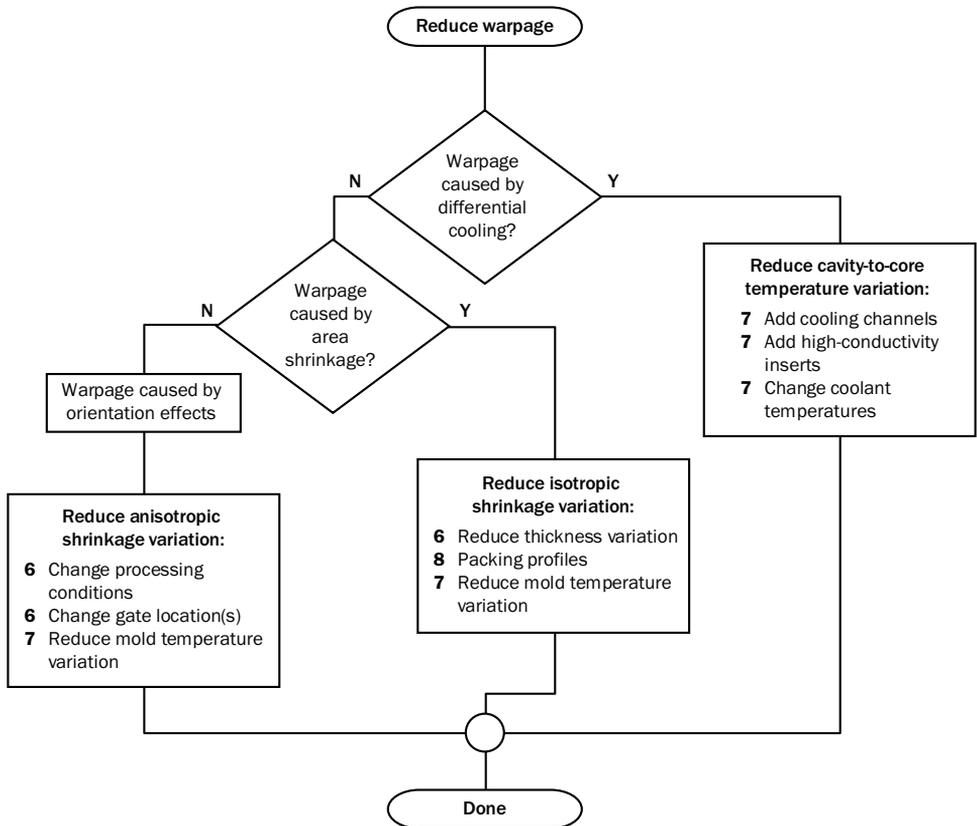


Figure 11.9 Reducing warpage

11.3 Using Moldflow to Evaluate an Initial Design

11.3.1 Description of this Example

The part for this example is a cover. The initial design, shown in Figure 11.10, is based on a previous year's design. The part has more rigorous specifications than in previous years. In this phase, the initial design will be evaluated to determine if further action is necessary.

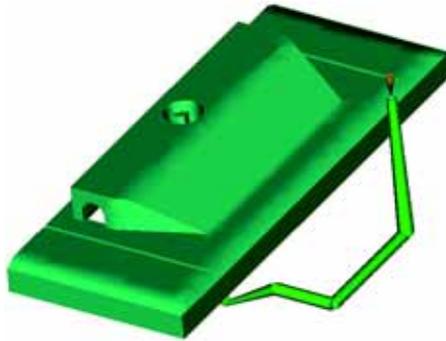


Figure 11.10 Original cover design

11.3.1.1 Specifications

The cover will be made in a two-plate, two-cavity tool, using a molding machine with a 100-tonne clamp force, and a 140 MPa pressure limit. Only one cavity is modeled because the second cavity is symmetric to the first cavity. The symmetry is accounted for by a technique called occurrence numbers. The material is a 33% glass-filled nylon 6. With the redesign for a new model year, the flatness of the bottom edge must be within 1.0 mm because of assembly requirements. The initial gate location is a submarine gate into a pin on the underside of the part. If necessary, other gate locations around the perimeter of the part may be used.

11.3.2 Molding Window

A molding window analysis was done on the cover, excluding the runner system. The results from the analysis are shown in Figure 11.11. They indicate that the processing window is wide with regard to mold and melt temperature ranges. The pressure to fill is well below the limit of the machine. The injection time range is about one second wide. Although it would be nice to have a wider range of injection times, a one-second range is very acceptable. The molding conditions picked are:

- Mold temperature: 80°C (176°F)

- Melt temperature: 280°C (536°F)
- Injection time: 0.9 seconds

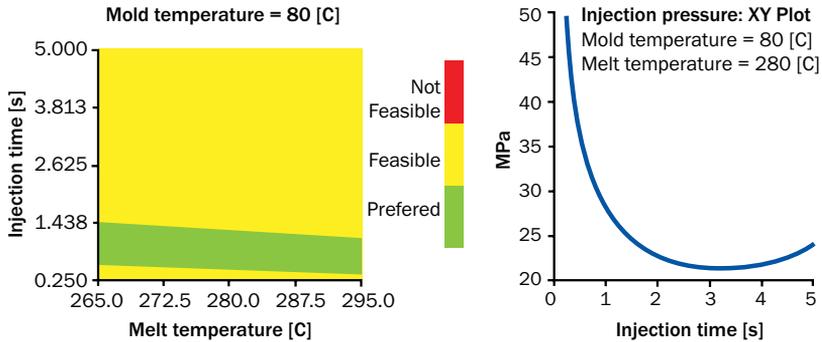


Figure 11.11 Molding window on the cover

11.3.3 Filling Analysis

Because the gate location for the initial analysis was fixed from a previous design, the gate location cannot be optimized for now. The gate location does not produce a balanced fill pattern. The part to the left of the gate, as shown in Figure 11.12, is overpacked. In addition to being overpacked, this area has a lower temperature and thicker frozen layer. The flow front velocity is not uniform, and could be addressed with a velocity profile if necessary. Other results did not show any problems other than the ones discussed above.

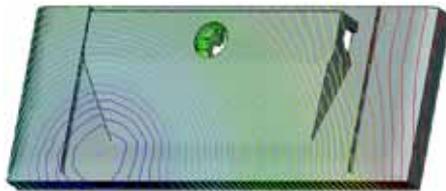


Figure 11.12 Cover fill time plot

11.3.4 Gate and Runner Design

The original gating system has a submarine gate into a pin. The diameter of the gate was 1.0 mm. With the flow rate required to fill the part in 0.9 second, the shear rate in the gate is very high, over 181,000 1/sec., as shown in Figure 11.13. The shear rate limit for nylon is 60,000 1/sec. However, because of the glass fillers in the material, the shear rate should be

well under the 60,000 1/sec. limit. The gate was opened up to 2.25 mm to reduce the shear rate to under 28,000 1/sec. For a glass-filled material, the shear rate could go a little lower, but it is not bad.

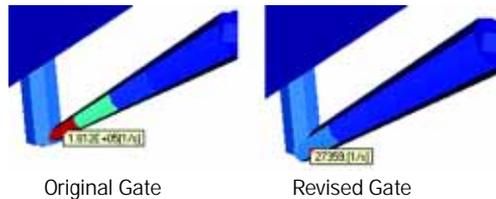


Figure 11.13 Cover gate shear rates

With the original gate and runner system, the shear heat in the runners was over 20°C (36°F). The original analysis used the optimized melt temperature as a starting point. Subsequent analyses lowered the melt temperature entering the sprue, so by the time the flow front got to the part the melt temperature would be about 280°C (536°F). The revised analysis lowered the melt temperature to 267°C (513°F), so the temperature entering the part was about the optimized temperature.

The cooling time of the feed system was compared to the part. The base of the sprue has the longest cooling time and is about 19 seconds, compared to the about four seconds for most of the part. However, there is a heavy boss on the cover that cools in about 12 seconds. The guideline says that the maximum runner cooling time should be no more than about two times that of the part. Considering there is a heavy section on the part that takes 12 seconds to cool, and the sprue cools in less than 24 seconds, the sprue cooling time is not unreasonable. Also, the sprue's orifice is the smallest standard commonly used, and it would not be practical to make it smaller. In addition, since the material is a glass-filled nylon, a smaller percentage of the cross section needs to be frozen for the part to be ejectable.

11.3.5 Cooling System Design

The cooling system for the cover consists of one circuit for the cavity and core, each having four lines. In the final design, the only thing that changed was a BeCu insert, which was added in the core to extract heat better. Figure 11.14 shows an example of the revised cooling system.

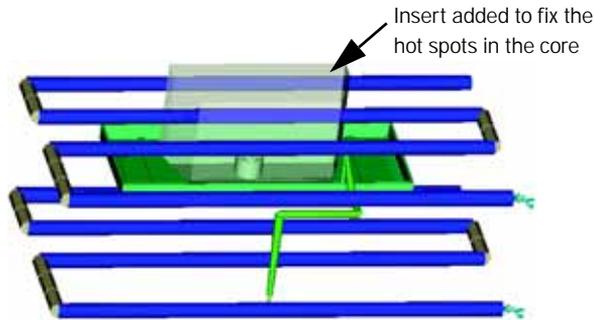


Figure 11.14 Revised cooling system

11.3.5.1 Mold Surface Temperature

The mold surface temperature of a part should be as uniform as possible. For semicrystalline materials this should be $\pm 5^\circ\text{C}$ from the target temperature, in this case 80°C (176°F). This is normally very difficult to achieve. The original design overcools much of the cavity, and the core runs very warm, up to 124°C (255°F). The main problem is the core is the narrow space between the boss and side wall. This narrow area adds 15°C to the core. A BeCu insert was used in the core to help extract the heat more efficiently. The core is now quite uniform in temperature. More than 50% of the surface area of the part is now within the tight $\pm 5^\circ\text{C}$ tolerance, and less than 10% is outside $\pm 10^\circ\text{C}$ tolerance, mostly to the cold side.

For both the original and revised designs, the coolant temperatures were adjusted to ensure the average mold surface temperature was close to the target temperature. In the case of the original design, the inlet temperatures were different for the cavity and core circuits. This is not always practical. With the revised design, the coolant temperatures were the same.

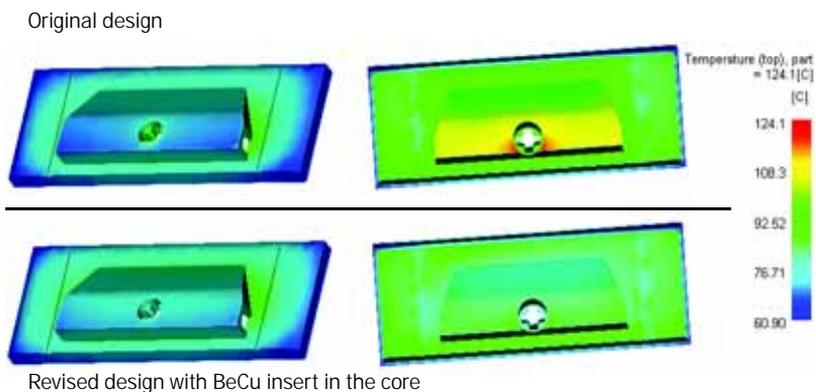


Figure 11.15 Mold surface temperature for the cover

The improvement in the temperature uniformity can also be seen in the temperature profile of the part. In Figure 11.16, two locations were tracked through the thickness of the part. One location (L1) is where the core runs very hot, and the other location is near the middle of the part (L2). The core side at both locations 20 to 35°C hotter in the original design than the revised design. Because the cavity side coolant line design was not modified, the temperatures on the cavity side are quite similar.

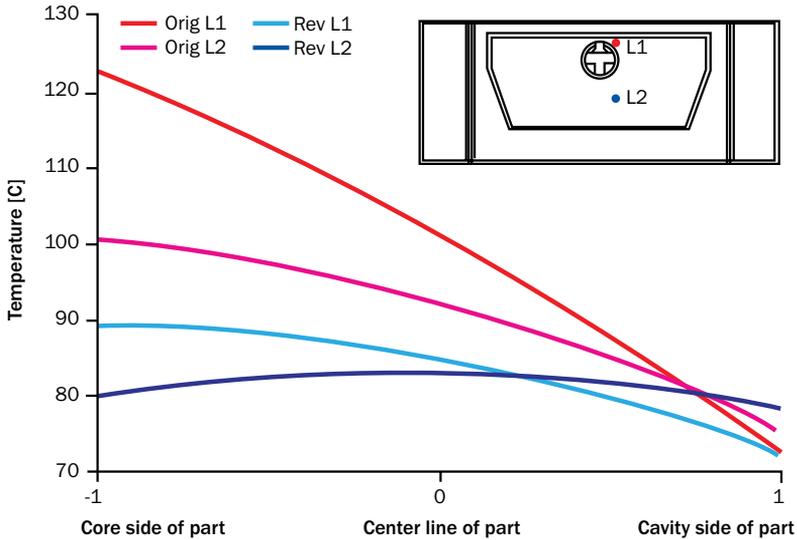


Figure 11.16 Temperature profile of the original and revised cooling designs at two locations on the part

11.3.6 Packing Analysis

The packing analysis was run to achieve a reasonable range of volumetric shrinkage without a significant amount of optimization. The material used is glass-filled, and the orientation of the fibers normally dictates the warpage. If the warpage is out of tolerance and the cause of warpage is differential shrinkage, then the packing can be optimized. The range of volumetric shrinkage is 3 to 5.1%, which is a good range.

11.3.7 Warpage Analysis

Warpage analysis, which requires input from a packing analysis, can now be run because the packing analysis has finished. The primary concern with the warpage analysis is the flatness of the bottom edge of the part. The edge must be flat within 1.0 mm (0.039 in). The warpage results were set up to make it easier to determine the flatness. Figure 11.17 shows the warpage

of the bottom edge exaggerated 10 times to help show the change in shape and the magnitude of the warpage. Three of the four corners were anchored to form a reference plane to measure the deflection. The edge warped about 1.5 mm (0.059 in). This is 1.5 times the tolerance.

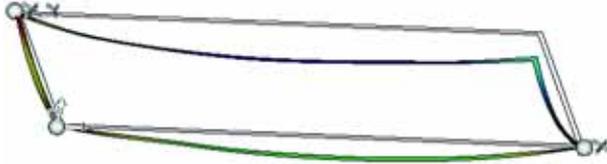


Figure 11.17 Bottom edge warpage of the cover's original design; the warpage magnitude is exaggerated 10 times to better show the change in shape

11.4 Using Moldflow to Optimize the Design

11.4.1 Determine the Cause of Warpage

Now that warpage analysis indicates the cover is well outside the flatness tolerance, the warpage must be fixed. Before the warpage can be fixed, the cause of warpage must be run to determine the cause.

For many parts, there are a combination of causes and offsetting effects. For example in some parts, the primary cause of warpage is differential shrinkage, but differential cooling is offsetting some of the differential shrinkage, so the total warpage is less than the differential shrinkage.

In the case of the cover, virtually all of the warpage is caused by orientation effects. Both differential cooling and differential shrinkage have slight offsetting effects.

11.4.2 Investigating Different Gate Locations

For fiber-filled materials, orientation effects could mean molecular orientation or fiber orientation. Fiber orientation dominates over molecular orientation. How fibers are oriented is very complex. Orientation can change through the thickness from highly oriented in the direction of flow, to random, to highly oriented across the direction of flow. Radial flow fronts tend to orient fibers across the flow direction, and shear rate tends to orient fibers in the direction to flow.

The gate location has a large influence on fiber orientation because of its influence on the shape of the flow front. Several different gating locations were investigated around the edge of the part.

Figure 11.18 shows how three different gate locations on the edge of the part compare to the original gate location. The gate location in the center of the long side make the warpage worse. With one or two gate locations on the narrow side of the part, the warpage is reduced, but not below the 1.0 mm tolerance. The different gate locations do not include runner systems, and the processing conditions are the same. The injection time could be optimized for the gate locations possibly improving the gate locations. However, other alternatives need to be investigated.

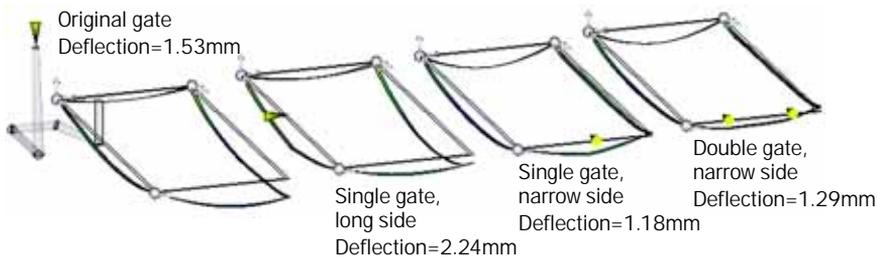


Figure 11.18 Cover comparing gate locations and deflections of the bottom edge of the part

Initially ruled out as a possibility, a gate location in the center of the part was tried. Figure 11.19 shows the gate location and resulting warpage of the part's edge. This shows that the warpage of the edge is now under the tolerance. By changing the gate location, the tool design needs to change from a two-plate tool with cold runners to a two-plate tool with a hot drop. Done early enough in the design phase, changing the tool configuration is not a significant issue, and the benefits can be easily shown using Moldflow.



Figure 11.19 Cover deflection with a gate on the top of the part

11.4.3 Validating the Best Gate Location

The different gate locations were investigated without a runner system attached. Now that the gate location has been moved, the feed system for the optimized gate location has to be designed.

11.4.3.1 Re-design the Cavity Cooling Lines

Because the gate location was moved to the center of the part, the coolant line locations needed to be re-designed because of the hot drop location on the part, as shown in Figure 11.20. Primarily, the lines across the part were oriented in the y-direction rather than the x-direction.

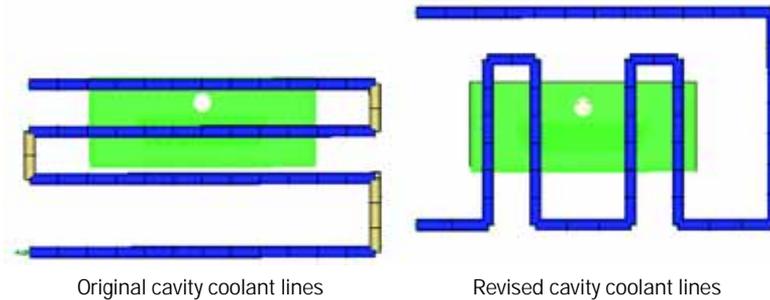


Figure 11.20 Original and revised cavity coolant line locations

11.4.3.2 Design a Hot Drop

With the location of the gate set, another molding window analysis was done to validate the injection time. Because the flow length was shorter, the injection time should be a little faster. Even though the original 0.9 seconds will work, 0.7 seconds was found to be a bit better. The orifice for the hot drop was initially set at 2.0 mm. The first analysis indicated that the shear rate was nearly 80,000 1/sec., which is above the shear rate limit for the material. The warpage analysis indicated that the warpage was lower with the faster injection time.

The gate was opened to 2.5 mm. The flow rate was also increased to make the nominal injection time 0.6 seconds. The warpage dropped again. The shear rate was also lowered to 53,000 1/sec. This shear rate should be lower because of the glass fillers, but the orifice is larger than the nominal wall of the part. The hot drop vendor should be consulted to ensure the gate orifice can be made to the desired size without leaving a large mark on the part.

11.4.3.3 Warpage with the Runner System

Once the hot drop was sized, a packing analysis was done so the volumetric shrinkage was about the same as the analysis with the injection location directly on the part. With the changes in the cooling and filling the warpage was better than just the gate location on the part. A comparison of the original gate location and the final is shown in Figure 11.21.

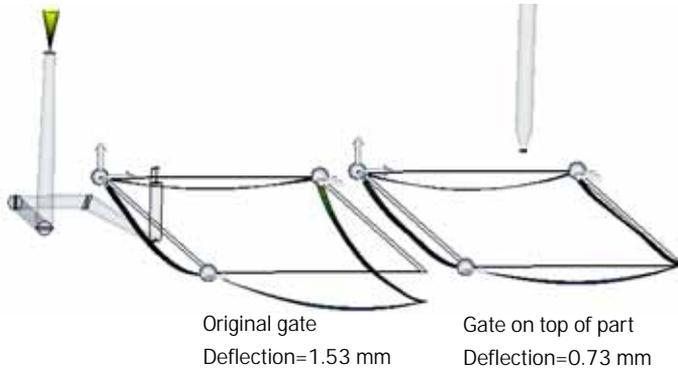


Figure 11.21 Warpage of bottom edge of the part with original and final gate locations

12 Part Defects

- Air traps
- Black specks and black streaks
- Brittleness
- Burn marks
- Delamination
- Dimensional variation
- Discoloration
- Fish eyes
- Flash
- Flow marks
- Hesitation
- Jetting
- Ripples
- Short shot
- Silver streaks
- Sink marks and voids
- Weld lines and meld lines

12.1 Air Traps

12.1.1 What Is an Air Trap?

An *air trap* is air caught inside the mold cavity. It becomes trapped by converging polymer melt fronts or because it failed to escape via the mold vents, or mold inserts, which also act as vents. Air-trap locations are usually in areas that fill last. Lack of vents or undersized vents in these last-to-fill areas are a common cause of air traps and the resulting defects. Another common cause is racetracking (the tendency of polymer melt to flow preferentially in thicker sections), caused by a large thickness ratio.

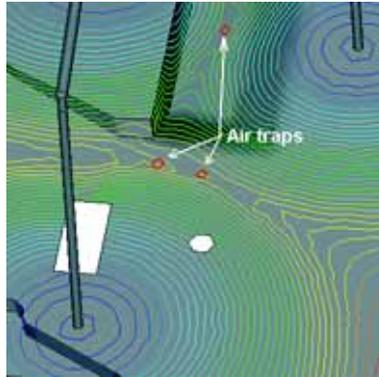


Figure 12.1 Air trap locations indicated by the air trap and fill time simulation results.

12.1.2 Problems Caused by Air Traps

Trapped air will result in voids and bubbles inside the molded part, a short shot (incomplete fill), or surface defects such as blemishes or burn marks.

12.1.3 Remedies

12.1.3.1 Alter the Part Design

Reducing the thickness ratio will minimize the racetracking effect of polymer melt.

12.1.3.2 Alter the Mold Design

Pay close attention to the proper placement of your vents. Place vents in the areas that fill last. Vents are typically positioned at discontinuities of mold material, such as at parting surfaces, between the insert and mold wall, at ejector pins, and at mold slides.

Re-design the gate and delivery system. Changing the delivery system can alter the filling pattern in such a way that the last-to-fill areas are located at the proper venting locations.

Make sure the vent size is large enough so that the air present in the cavity can escape during injection. Be careful, however, that the vent is not so large that it causes flashes in the vent. The recommended vent size is 0.025 mm (0.001 in) for crystalline polymers, and 0.038 mm (0.0015 in) for amorphous polymers.

12.1.3.3 Adjust the Molding Conditions

Reduce the injection speed. High injection speeds can lead to jetting, which causes air to become entrapped in the part. Lowering the injection speed will give the air displaced by the melt sufficient time to escape from the vents.

12.2 Black Specks and Black Streaks

12.2.1 What Are Black Specks and Black Streaks?

Black specks and *black streaks* are dark spots or dark streaks found on the surface or throughout a molded part. Brown specks or streaks refer to the same type of defect, except the burning or discoloration is not as severe.

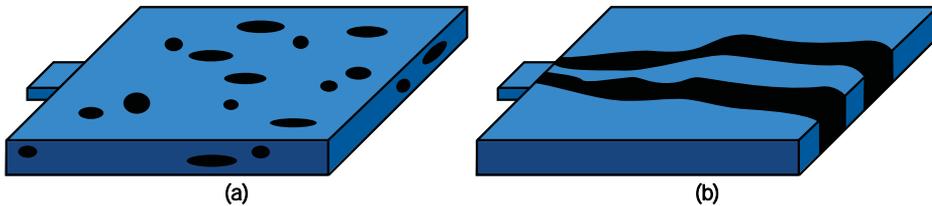


Figure 12.2 (a) Black specks and (b) Black streaks

12.2.2 Causes of Black Specks and Black Streaks

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

12.2.2.1 Material Degradation

Overheated materials can degrade and lead to black streaks. Material that stays in the nicked rough surfaces of the barrel wall and screw surfaces for a prolonged period of time after heating will char and degrade, resulting in the defect.

12.2.2.2 Material Contamination

Contaminants in the air or material, such as dirty regrind, foreign material, different color material, or a lower melt-temperature material, are what most often lead to black specks and black streaks. Airborne dirt can also cause dark spots on the surface of a molded part.

12.2.2.3 Other Defects Resulting from the Same Causes

- Brittleness
- Burn marks
- Discoloration

12.2.3 Remedies

12.2.3.1 Handle the Material Carefully

Make certain no contaminated materials, such as dirty regrind, are blended into the original material. Put the cover on the hopper and all bins of material. Airborne dirt can contaminate the original material, leading to black spots.

12.2.3.2 Alter the Mold Design

Clean the ejectors and slides. The streaks could 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.

12.2.3.3 Select a Proper Machine

Size a proper injection machine for a specific mold. The typical shot size should be between 20 and 80% of machine injection capacity. For temperature-sensitive materials, the range should be narrowed down more. Plastics simulation software can help you select the right size injection machine for a specific mold. This will help avoid resin remaining in the heated barrel for prolonged periods of time.

Check for scratched or dented barrel/screw surfaces that trap material. This could lead to the material becoming overheated or burned.

Check for local overheating by a run-away heater band or a malfunctioning temperature controller.

12.2.3.4 Adjust the Molding Conditions

Lower the barrel and nozzle temperature. Material degradation can result from a high melt temperature.

Purge and clean the injection unit. The black streaks might be caused by contamination from the barrel wall or the screw surface. When molding with two materials, after switching from one material to the other, the old material might not be purged from the barrel completely. This could generate defects during the molding of the second material.

Avoid recycling rejected parts with black specks and black streaks. Recycling such parts could lead to further contamination, unless they will be used for parts that are in black or for which such defects are acceptable.

12.3 Brittleness

12.3.1 What Is Brittleness?

A brittle molded part has a tendency to break or crack. *Brittleness* results from material degradation leading to shorter molecular chain length (thus lower molecular weight). As a result, the physical integrity of the part is substantially less than the specification.

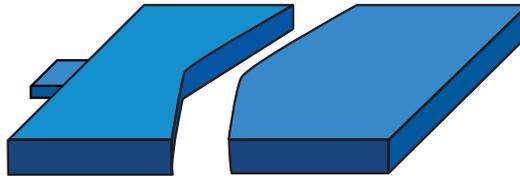


Figure 12.3 Degraded part tends to be brittle and break easily

12.3.2 Causes of Brittleness

Brittleness is caused by material degradation because of:

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

12.3.2.1 Other Defects Resulting from the Same Causes

- Black specks and black streaks
- Burn marks
- Discoloration

12.3.3 Remedies

12.3.3.1 Adjust the Material Preparation

Set proper drying conditions before molding. Brittleness can be caused by excessive drying time or drying temperature such as at full heat for several days. Excessive drying either drives

off volatiles in the plastic, making it more sensitive to processing, or degrades the material by reducing the molecular weight. Material suppliers can provide optimum drying conditions for the specific materials.

Reduce regrind material. The brittleness could be caused by too much reground material added into the original virgin material.

Change to a high-strength material since low-strength materials tend to become more brittle if processed improperly.

12.3.3.2 Alter the Mold Design

Enlarge the sprue, runner, and/or gate. Restrictive sprue, runner, gate, or even part design could cause excessive shear heating that aggravates an already overheated material, causing material degradation.

12.3.3.3 Select a Proper Machine/Machine Component

Get a better screw design for the material being used to achieve a better mixed melt temperature. Contact material suppliers to get the right screw design information to avoid improper melt mix or overheating that leads to material degradation.

12.3.3.4 Adjust Molding Conditions

Reduce the barrel temperature and nozzle temperature. If the barrel and nozzle temperature are too high, the material in the barrel will be overheated, leading to thermal degradation and the color change.

Reduce the back pressure, screw rotation speed, or injection speed because shear heating can result in material degradation.

While not overheating the material, increase melt temperature, mold temperature or injection pressure if the weld line has a tendency to crack. See Section 12.17 for more information about weld lines and meld lines.

12.4 Burn Marks

12.4.1 What Is a Burn Mark?

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



Figure 12.4 Burn marks

12.4.2 Causes of Burn Marks

12.4.2.1 Entrapped Air

If the injection speed or injection pressure is too high, the air trapped in the runner system and cavity cannot be released to the atmosphere through the venting system properly within a very short filling time. Air traps also occur in improperly vented systems when race-tracking behavior is significant. Consequently, the air will be compressed, resulting in a very high pressure and temperature, which will cause the polymer to degrade on the surface near the end of the flow path or the blind area.

12.4.2.2 Material Degradation

Burn marks can also result from the degraded (charred) materials being carried downstream and then appearing on the surface of the molded part or near the venting areas. Material degradation is caused by:

- High melt temperature: Excessive melt temperature can be caused by improper barrel temperature setting, a broken thermocouple, or a malfunctioning temperature controller.
- High screw-rotation speed: If the screw speed is too high during the plasticization period, it will create too much frictional heat, which could degrade the material.
- Restrictive flow path: When the melt flows through restrictive nozzle, runner, gate, or part sections it creates a lot of shear (frictional) heat, which could degrade the material.

12.4.2.3 Other Defects Resulting from the Same Causes

- Black specks and black streaks
- Brittleness
- Discoloration

12.4.3 Remedies

12.4.3.1 Alter the Mold Design

Place an adequate venting system throughout the mold to help the entrapped air escape. Vents are especially important near the end of the flow path and in the blind area. The recommended venting size is 0.025 mm (0.001 in) for crystalline polymers and 0.038 mm (0.0015 in) for amorphous polymers.

Enlarge the sprue, runner, and/or gate. Restrictive sprue, runner, gate, or even part design could cause excessive shear heating that aggravates an already overheated material, causing material degradation.

12.4.3.2 Adjust the Molding Conditions

Reduce the likelihood of burn marks by avoiding excessive melt temperatures during the molding process:

- Reduce the injection pressure
- Reduce the injection speed
- Reduce the screw rotation speed
- Decrease the barrel temperature
- Check the band heaters on the barrel and nozzle, and calibrate the thermocouple

12.5 Delamination

12.5.1 What Is Delamination?

Delamination (sometimes called *lamination* or *layering*) is a defect in which the surface of a molded part can be peeled off layer by layer.

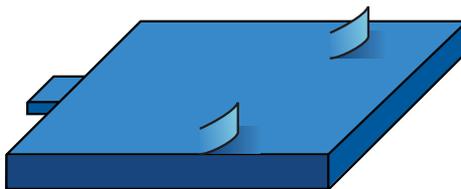


Figure 12.5 Delamination causes layer-wise peel-off on the surface of a molded part

12.5.2 Causes of Delamination

Delamination can be caused by several factors, including:

- Incompatible materials blended together
- Too much mold release agent being used during the molding process
- Low melt temperature in the cavity
- Excessive moisture
- Sharp corners at the gate and runner

12.5.3 Remedies

12.5.3.1 Change the Material Preparation

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

12.5.3.2 Alter the Mold Design

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

12.5.3.3 Adjust the Molding Conditions

Avoid using excessive mold release agent to fix the de-molding problem. Delamination can be caused by excessive use of mold release agent. You should repair the ejection system or other problems to eliminate the difficulty of de-molding instead of overusing the mold release agent.

Follow the pre-dry instructions for the specific material and pre-dry the material properly before molding. Excessive moisture heats up and forms steam, which results in delamination on the surface.

Increase the barrel temperature and mold temperature. If the melt temperature is too low, layers of material are formed because they can't bond to each other. When ejected or subjected to stress, they separate from each other.

12.6 Dimensional Variation

12.6.1 What Is Dimensional Variation?

Dimensional variation is a defect characterized by the molded part dimension varying from batch to batch or from shot to shot while the machine settings remain the same.

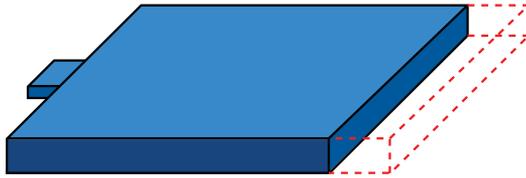


Figure 12.6 Dimensional variation is an unexpected change of part dimension

12.6.2 Causes of Dimensional Variation

Dimensional variation can be caused by:

- Unstable machine control
- A narrow molding window
- Improper molding conditions
- A broken check ring (within the injection unit)
- Unstable material properties

12.6.3 Remedies

12.6.3.1 Improve the Material Preparation

Contact the material vendor and change the material lot if the material has batch-to-batch variation.

Pre-dry the material before molding if the material is too wet.

Limit the percentage of regrind material added to the origin material. The irregular particle size can cause different levels of mixed melt material, and lead to unstable molded part dimensional variation.

12.6.3.2 Change a Mold Design or Component

Fix or adjust the ejection system if the molded part is bowed or distorted during ejection.

Design a proper runner and gate system for a specific mold and material. Use plastic injection molding simulation software to optimize the runner system dimensions to assure a smooth melt flow into the cavity.

12.6.3.3 Change a Machine Component

Replace the check ring if it is broken or worn out.

Replace heater bands or the thermocouple if it is out of order and causes unstable melt flow.

12.6.3.4 Adjust the Molding Conditions

Increase the injection and packing pressure. Make sure enough material is delivered into the cavity during the filling and packing stages.

Increase the injection and packing time to be sure enough material is delivered into the cavity during filling and packing stages.

Make sure the mold temperature is even by checking the cooling system.

Set up screw metering and injection stroke, screw-rotation speed, and back pressure properly so that they fall within the process window.

12.7 Discoloration

12.7.1 What Is Discoloration?

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

12.7.2 Causes of Discoloration

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

- The material stays in the barrel too long
- The barrel temperature is too high, causing the color to change
- Contamination was caused by reground material, different color material, or foreign material

12.7.2.1 Other Defects Resulting from the Same Causes

- Black specks and black streaks
- Brittleness
- Burn marks

12.7.3 Remedies

12.7.3.1 Handle the Material Carefully

Maintain proper housekeeping for virgin material and regrind material storage to avoid contamination of the materials.

12.7.3.2 Alter the Mold Design

Add an adequate venting system. To avoid discoloration (or burn marks) caused by poor venting or air traps, use the recommended venting size: 0.025 mm (0.001 in) for crystalline polymers and 0.038 mm (0.0015 in) for amorphous polymers.

12.7.3.3 Select a Proper Machine

Use a different size injection-molding machine. The typical shot size should be between 20 and 80% of machine injection capacity. For temperature-sensitive materials, the range should be narrowed down, depending on the material. Plastics simulation software can help you select the right size machine for a specific mold. This will help avoid the resin remaining in the heated barrel for prolonged periods of time.

12.7.3.4 Adjust the Molding Conditions

Clean the hopper completely. It is important to avoid foreign material or different color materials mixing together before molding.

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

Reduce the barrel and nozzle temperatures. If the barrel and nozzle temperatures are too high, the material in the barrel will be overheated, leading to thermal degradation and the color change.

12.8 Fish Eyes

12.8.1 What Are Fish Eyes?

Fish eyes are a surface defect that results from unmelted materials being pushed with the melt stream into the cavity and appearing on the surface of a molded part.

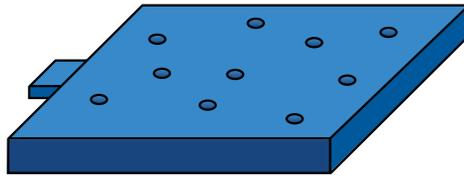


Figure 12.7 Unmelted materials in the melt stream causes fish eyes

12.8.2 Causes of Fish Eyes

Fish eyes are caused by:

- **Low barrel temperature:** If the barrel temperature is too low to melt the materials completely, the unmelted pellets will merge with the melt stream, marring the surface of the part.
- **Too much regrind:** The shape and size of regrind is irregular compared with original material, which can trap more air and cause the material to blend unevenly.
- **Material contamination:** If a high-melt-temperature material is blended into the original material, the blended material may stay in pellet form and cause fish eyes during the molding process.
- **Low screw rotation speed and back pressure:** If the screw rotation speed and the back pressure settings are too low, there might not be enough frictional heating to melt the material completely in the barrel before injection.

12.8.3 Remedies

12.8.3.1 Improve the Material Preparation

Limit or eliminate regrind for practical molding, depending on part quality requirements. Adding 10% of regrind is a good start, if regrind is allowed.

Store different materials separately and keep covers on the containers or bags to avoid blending different materials.

12.8.3.2 Adjust the Molding Conditions

Material suppliers usually provide the information about barrel temperature, back pressure, and screw rotation speed for specific materials. If you have followed suppliers' recommendations and are still experiencing problems, try making the following adjustments:

- Increase the barrel temperature
- Increase the back pressure to blend melt materials evenly
- Increase the screw rotation speed during the plasticization stage to create more frictional heat to melt materials

12.9 Flash

12.9.1 What Is Flash?

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

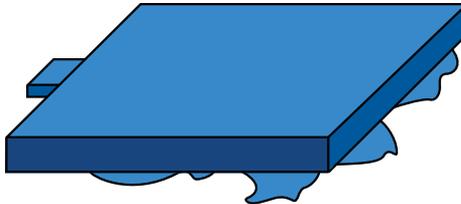


Figure 12.8 Flash

12.9.2 Causes of Flash

12.9.2.1 Low Clamp Force

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

12.9.2.2 Gap within the Mold

Flash will occur if the parting surface does not contact completely, due to a deformed mold structure, parting surface defect, improper machine and mold set up, or flash or foreign material stuck on the parting surface.

12.9.2.3 Molding Conditions

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

12.9.2.4 Improper Venting

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

12.9.3 Remedies

12.9.3.1 Adjust the Mold Set-up

Set up the mold to seal properly. A mismatch or undesirable gap between the cavity and core sides of the mold will result in flash.

Make sure the mold plates are strong enough to avoid deformation during molding. Add pillar support or thicken the mold plates if there is any deformation of the mold plate during the molding process.

Check for adequate venting dimensions.

The recommended venting size is 0.025 mm (0.001 in) for crystalline polymers and 0.038 mm (0.0015 in) for amorphous polymers.

Clean the mold surface. Flash can be caused by the mold surface not sealing well due to foreign material remaining between the parting surfaces.

Mill out the surface to keep the sealing pressure of land area around the cavities high enough.

12.9.3.2 Adjust the Machine Settings

Set up the machine and mold to seal properly. Flash can be caused by a poor seal between the cavity and core sides of the mold, and machine platens that are not parallel.

Increase the injection molding machine size. Flash can result from insufficient machine clamp force.

Adjust the clamp force if the machine capacity does not have enough clamp force.

12.9.3.3 Adjust the Molding Conditions

Decrease the barrel temperature and nozzle temperature. A high melt temperature reduces the melt viscosity, making a thinner melt, which causes flash. But beware: avoid melt temperatures so low such that the resulting high injection pressure required causes flash.

Reduce the injection and packing pressure to reduce the clamp force requirement.

Reduce the feed setting (stroke length) to reduce metering (over-fill).

Increase the injection time or reduce the injection speed.

12.10 Flow Marks

12.10.1 What Is A Flow Mark?

A *flow mark* or *halo* is a surface defect in which circular ripples or wavelets appear near the gate.



Figure 12.9 Flow mark

12.10.2 Causes of Flow Marks

Flow marks are caused by cold material near the gate or lack of compensated material during the packing stage. The problem usually can be attributed to:

- Low melt temperature
- Low mold temperature
- Low injection speed
- Low injection pressure
- Small runner stem and gate

According to a recent visual analysis using a glass-inserted mold, the flow mark defect can also be caused by cooling of the flow front portion on a cavity wall and the repeated phenomena of “getting over” and cooling with the subsequent melt. See Section 12.13 for more information about ripples.

12.10.3 Remedies

12.10.3.1 Alter the Mold Design

Change the size of the cold well in the runner system to trap the cold material during the filling stage. The proper length of the cold well is usually equal to that of the runner diameter.

Increase the runner system and gate size for the specific mold and material. Flow marks are sometimes caused by a restrictive runner system and gate size that freeze-off prematurely so that the material cannot be compensated during the packing stage.

Shorten the sprue length or use a hot runner design instead of a cold runner design.

12.10.3.2 Adjust the Molding Conditions

Increase the injection pressure and packing pressure.

Increase the barrel and nozzle temperature.

Increase the mold temperature.

12.11 Hesitation

12.11.1 What Is Hesitation?

Hesitation (or a *hesitation mark*) is a surface defect that results from the stagnation of polymer melt flow over a thin-sectioned area, or an area of abrupt thickness variation. Hesitation can be eliminated by changing the part thickness or moving the gate location.

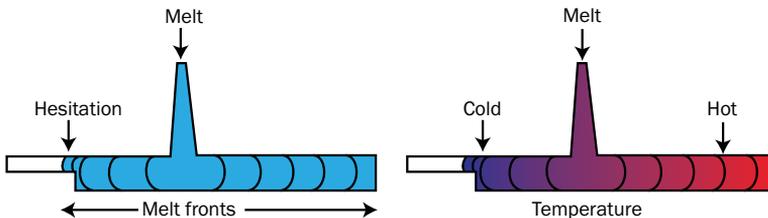


Figure 12.10 Hesitation results from stagnation of polymer melt flow

12.11.2 Problems Caused by Hesitation

When polymer melt is injected into a cavity of variable thickness, it tends to fill the thick and less resistant areas. As a result, polymer melt may stagnate at thin sections until the rest of the part is filled and the stagnated polymer melt starts moving again (see Figure 12.10). However, if the duration of hesitation is significant, polymer will solidify prematurely at the stagnated point. When the solidified melt front is pushed to the part surface, a surface defect such as a hesitation mark occurs.

12.11.3 Remedies

When troubleshooting the cause of hesitation in your part, both the part and mold design will need to be re-examined. Fine-tuning the processing conditions should also be tried.

12.11.3.1 Alter the Part Design

Reduce part thickness variation by changing wall thicknesses, if possible.

12.11.3.2 Alter the Mold Design

Position the gate away from the thin-sectioned areas, or regions of sudden thickness change. In this manner, hesitation will occur at a later time, and for a shorter duration. Figure 12.11 shows that in a poor design, hesitation results from stagnation of polymer melt flow; moving the gate away from the thin section reduces hesitation.

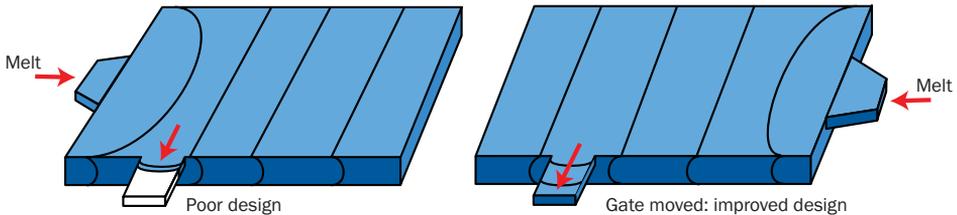


Figure 12.11 Hesitation resulting from improper gate position

12.11.3.3 Adjust the Molding Conditions

Increase the melt temperature and/or the injection pressure.

12.12 Jetting

12.12.1 What Is Jetting?

Jetting occurs when polymer melt is pushed at a high velocity through restrictive areas, such as the nozzle, runner, or gate, into open, thicker areas without forming contact with the mold wall. The buckled, snakelike jetting stream causes contact points to form between the folds of melt in the jet, creating small-scale “welds” (see Figure 12.12).

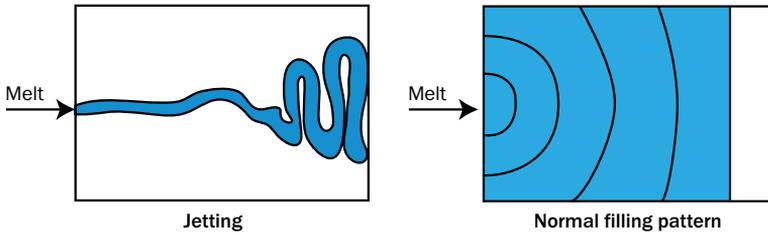


Figure 12.12 Jetting vs. normal filling pattern.

12.12.2 Problems Caused by Jetting

Jetting leads to part weakness, surface blemishes, and a host of internal defects. Contrast this with a normal filling pattern, in which melt advances in a progressive pattern from the gate to the extremities of the cavity, as illustrated in Figure 12.12.

12.12.3 Remedies

12.12.3.1 Alter the Mold Design

The trouble often lies with the gate design.

Direct the melt against a metal surface.

Use an overlap gate or a submarine gate as shown in Figure 12.13.

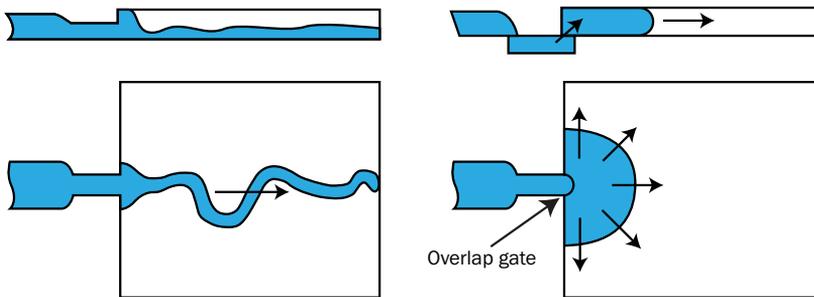


Figure 12.13 Using an overlap gate to avoid jetting

Slow down the melt with a gradually divergent flow area. A tab or fan gate provides a smooth transition from the gate to the cavity (Figure 12.14). This reduces the melt shear stress and shear rate.

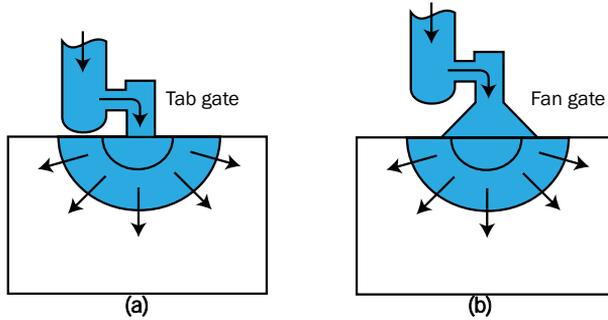


Figure 12.14 Using a tab gate (a) and a fan gate (b) to avoid jetting

Enlarge the size of the gate and runner or reduce the gate-land length. You can also relocate or redesign the gate in one of the following ways to reduce jetting.

12.12.3.2 Adjust the Molding Conditions

Adjust the ram-speed profile. Use an optimized ram-speed profile so that melt-front velocity is initially slow when the melt passes through the gate, then increases once a dispersed, tongue-shaped material is formed near the gate. Figure 12.15 illustrates this technique.

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

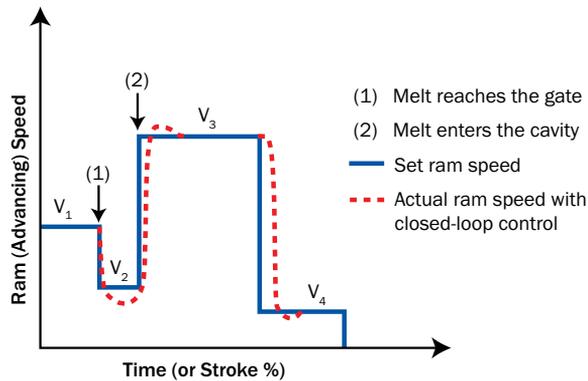


Figure 12.15 Adjust ram-speed profile to avoid jetting.

12.13 Ripples

12.13.1 What Are Ripples?

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



Figure 12.16 Ripples

12.13.2 Cause of Ripples

According to a recent visual analysis using a glass-inserted mold, the ripple defect is caused by the flow front portion of the melt cooling on a cavity wall, and the repeated phenomena of the subsequent melt “getting over” and cooling, as shown in Figure 12.17 below. Flow-front velocity and mold temperature have a stronger influence on the formation of ripples compared to the shape of the gates and the melt temperature.

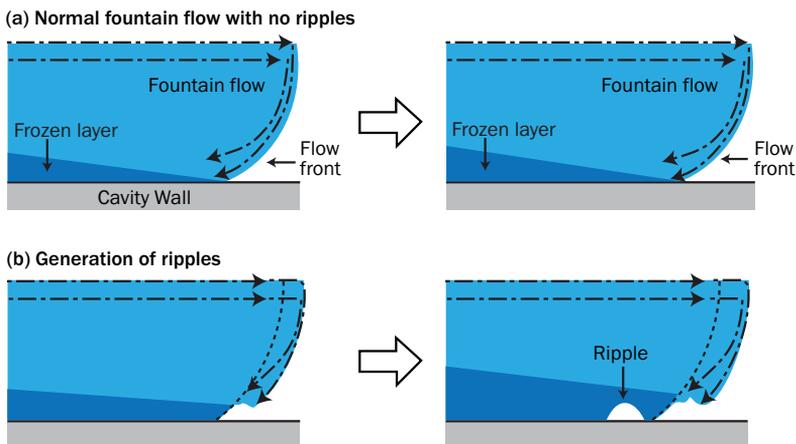


Figure 12.17 (a) Normal filling with no ripples (2) Generation of ripples with cold mold wall and low flow-front velocity

12.13.3 Remedies

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

12.13.3.1 Modify the Part Design

Increase the part thickness.

12.13.3.2 Change the Mold Design

Make sure the runner system (including the sprue, runners, and gates) is adequate for the specific part.

Place an optimal venting system throughout the entire mold, especially around the end of the flow path. Make sure the venting system is large enough that the air present in the cavity can escape during injection. Be careful, however, that the venting system is not so large that it causes flash at the edge of the molding. The recommended venting size is 0.025 mm (0.001 in) for crystalline polymers, and 0.038 mm (0.0015 in) for amorphous polymers.

12.13.3.3 Adjust the Molding Conditions

Increase the mold temperature.

Increase the injection speed. This will create more viscous heating and reduce the melt viscosity.

Increase the injection pressure. Be careful not to exceed the machine's capacity. The operating injection pressure should normally be limited to 70 to 85% of the maximum injection pressure to prevent accidental damage to the machine's hydraulic system.

Increase the melt temperature. Be careful not to introduce material degradation from prolonged exposure at an elevated temperature.

12.14 Short Shots

12.14.1 What Is a Short Shot?

A *short shot* is a molded part that is incomplete because insufficient material was injected into the mold. In some cases, short shots are intentionally produced to determine or visualize the filling pattern. But problematic short shots occur when the polymer melt cannot fill the entire cavity (or cavities). This most commonly occurs at thin sections or extremities.

12.14.2 Causes of Short Shots

Any factors that increase the resistance of polymer melt to flow or prohibit delivery of sufficient material into the cavity can cause a short shot. These factors include:

- Insufficiently-sized restrictive-flow areas, such as gates, runners, and thin walls
- Low melt and/or mold-wall temperatures
- A lack of vents to bleed the air trapped inside the cavity
- Insufficient machine injection pressure (resulting from high melt resistance and a restricted flow path), volume, and/or ram speed
- Machine defects such as an empty hopper, blocked feed throat, or a worn nonreturn (check) valve that causes loss of injection pressure or leakage of injection volume
- Premature solidification of the polymer melt due to hesitation, poor filling pattern, or prolonged injection time

12.14.3 Remedies

Several factors influence the polymer's ability to fill the entire cavity. Proper remedial actions can be taken when the cause of a short shot is pinpointed. Here are some suggestions.

12.14.3.1 Alter the Part Design

It is important to facilitate the flow of injected polymer melt; doing so can alleviate short shots. Strategically increase the thickness of certain wall sections (as flow leaders).

12.14.3.2 Alter the Mold Design

A properly designed delivery system (sprue, runner, and gate) will facilitate a more balanced filling pattern. If needed, modify your design in the following ways:

- Fill the thick areas before filling the thin areas. Doing so will avoid hesitation, which causes early solidification of polymer.
- Increase the number and/or size of gates to reduce the flow length.
- Increase the size of runner systems to reduce resistance.

Entrapped air inside the mold cavity (air traps) can also lead to short shots.

- Place vents at the proper locations, typically near the areas that fill last. This should help vent the displaced air.
- Increase the size and number of vents.

12.14.3.3 Adjust the Molding Conditions

Look closely at the factors that control how material is injected into the mold.

- Increase the injection pressure. Do not exceed the machine's capability. To prevent accidental damage to the machine's hydraulic system, limit the operating injection pressure to 70 to 85% of the maximum injection pressure.
- Increase the injection speed. Within the machine limits, this will create more viscous heating and reduce the melt viscosity.
- Increase the injection volume.
- Increase the barrel temperature and/or the mold-wall temperature. Higher temperatures will promote the flow of material through the cavity. Be careful to avoid material degradation due to prolonged exposure at an elevated temperature.

The molding machine might also be the culprit if you are experiencing problematic short shots.

- Check the hopper for sufficient material supply or a clogged feed throat.
- Inspect the non-return valve and barrel for excessive wear. Wear can lead to loss of injection pressure and leakage of injection volume.

12.15 Silver Streaks

12.15.1 What Are Silver Streaks?

Silver streaks are the splash appearance of moisture, air, or charred plastic particles on the surface of a molded part, which are fanned out in a direction emanating from the gate location.



Figure 12.18 Silver streaks

12.15.2 Causes of Silver Streaks

Silver streaks can be caused by:

- **Moisture:** Plastic materials absorb a certain degree of moisture during storage. If the material is not dried properly before molding, the moisture residing in the resin will turn into a steam during the injection process and splay on the surface of the molded part.
- **Air:** During the plasticization period, a certain amount of gas can be trapped and blended into the melt material. If the air does not escape during the injection process, it could splay out on the surface of the molded part.
- **Degraded (charred) plastic particles:** There are a several reasons degraded (charred) plastic particles will splay on the surface of a molded part:
 - **Material contamination:** When molding with two materials, as you switch from one material to another, the residual particles left in the barrel could be charred if the second material is being molded at a higher temperature. In addition, contaminated, rejected parts and regrind will re-contaminate virgin material in the next batch of molded parts.
 - **Barrel temperature:** An improper barrel temperature setting may degrade polymer molecules, and they will begin to char.
 - **Shot volume:** If the shot size is below 20% of the machine injection capacity, especially for temperature-sensitive materials, the melt resin will remain in the barrel too long and will begin to degrade.

12.15.3 Remedies

12.15.3.1 Handle the Material Carefully

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

12.15.3.2 Alter the Mold Design

Enlarge the sprue, runner, and/or gate. Restrictive sprue, runner, gate, or even part design could cause excessive shear heating that aggravates an already overheated material, causing material degradation.

Check for adequate venting dimensions. The recommended venting size is 0.025 mm (0.001 in) for crystalline polymers, and 0.038 mm (0.0015 in) for amorphous polymers.

12.15.3.3 Adjust the Molding Conditions

The following precautions will deter material from degrading during the process:

- Size a proper injection machine for a specific mold. The typical shot size should be between 20 and 80% of the machine injection capacity. For temperature-sensitive

materials, the range should be narrowed down, depending on materials. Plastics simulation software can help you select the right size injection machine for a specific mold. This will help to avoid a prolonged residence time for resin in the heated barrel.

- Fully purge the older material from the barrel if switching material from one to the other. Old material particles left behind could be charred.
- Increase the back pressure. This will help minimize air blending into the melt material.
- Improve the venting system. It is important to allow air and steam to escape easily.
- Decrease the melt temperature, injection pressure, or injection speed.

12.16 Sink Marks and Voids

12.16.1 What Are Sink Marks and Voids?

A *sink mark* is a local surface depression that typically occurs in moldings with thicker sections, or at locations above ribs, bosses, and internal fillets. A *void* is a vacuum bubble in the core.

12.16.2 Causes of Sink Marks and Voids

Sink marks and voids are caused by localized shrinkage of the material at thick sections without sufficient compensation when the part is cooling. A sink mark almost always occurs on a surface that is opposite to and adjoining a leg or rib. This occurs because of unbalanced heat removal or similar factors.

Factors that lead to sink marks and voids are:

- Low injection and packing pressure
- Short hold time or cooling time
- High melt temperature or mold temperature
- Localized geometric features

After the material on the outside has cooled and solidified, the core material starts to cool. Its shrinkage pulls the surface of the main wall inward, causing a sink mark. If the skin is rigid enough, as in engineering resins, deformation of the skin may be replaced by formation of a void in the core. Figure 12.19 illustrates this phenomenon.

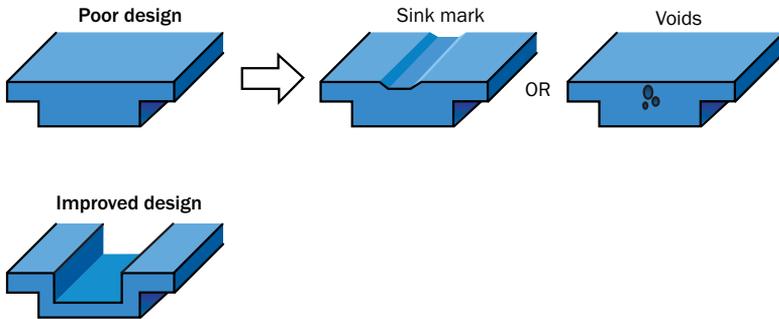


Figure 12.19 Sink marks and voids are created by material shrinkage without sufficient compensation

12.16.3 Remedies

Sink marks and voids can usually be alleviated by fine-tuning some combination of your part and mold design and the conditions under which the part is molded. Use the suggestions below to pinpoint and fix the problem.

12.16.3.1 Alter the Part Design

Conceal sink marks by adding a design feature, such as a series of serrations on the area where they occur. Figure 12.20 illustrates this technique.

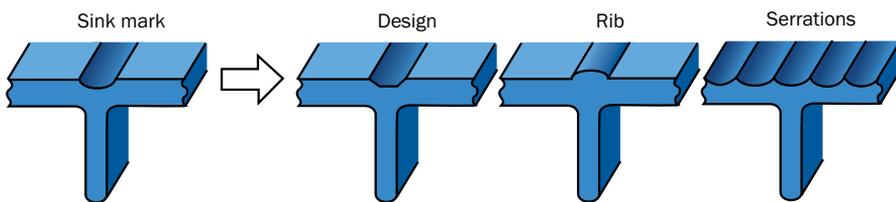


Figure 12.20 Sink marks can be eliminated by creating a design feature, rib, or serrations

Modify the part thickness design as suggested to minimize the thickness variation.

Re-design the thickness of the ribs, bosses, and gussets to be 50 to 80% of the attached (base) wall thickness. Figure 12.21 shows the dimensions we prescribe.

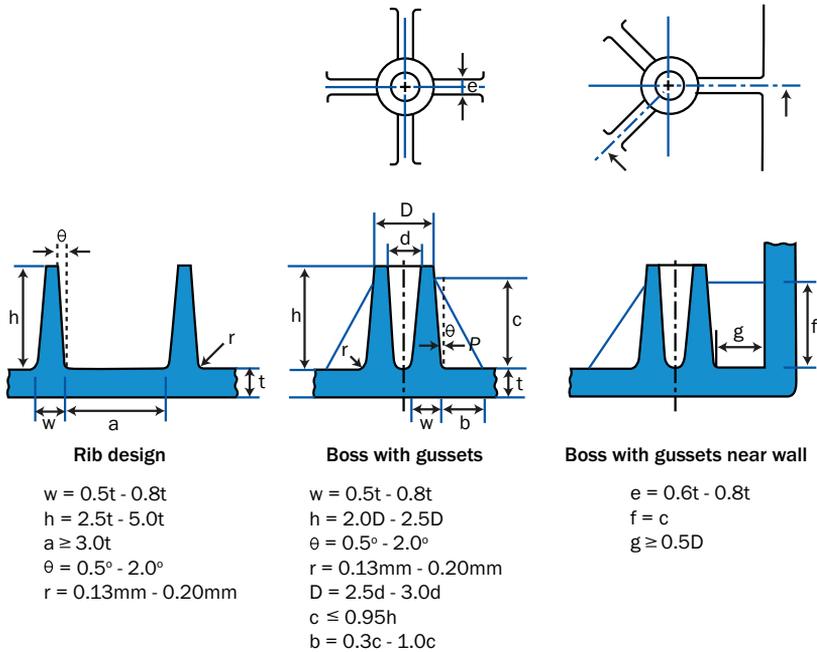


Figure 12.21 Recommended dimensions for ribs, bosses, and gussets

12.16.3.2 Alter the Mold Design

Increase the size of gates and runners to delay the gate freeze-off time. This allows more material to be packed into the cavity.

Add more vents or enlarge the existing vents. Vents allow air trapped inside the cavity to escape.

Relocate the gate to or near a thicker section. This allows them to be packed before the thinner sections freeze off.

12.16.3.3 Adjust the Molding Conditions

Increase the cushion at the end of the injection stroke. You should maintain a cushion of approximately 3 mm (0.12 in).

Increase the injection pressure and the holding time.

Increase the screw-forward time and decrease the injection rate.

Decrease the melt and mold-wall temperatures.

Increase the cooling time.

Check the non-return valve for possible material leakage.

12.17 Weld Lines and Meld Lines

12.17.1 What Are Weld Lines and Meld Lines?

A *weld line* (also called a *weld mark* or a *knit line*) is formed when separate melt fronts traveling in opposite directions meet. A *meld line* occurs if two emerging melt fronts flow parallel to each other and create a bond between them. Weld and meld lines can be caused by holes or inserts in the part, multiple gates, or variable wall thickness where hesitation or race tracking occurs. If weld or meld lines cannot be avoided, position them at low-stress and low-visibility areas by adjusting the gate position. Improve the strength of weld and meld lines by increasing the local temperature and pressure at their locations.

12.17.1.1 How to Tell the Difference between Weld Lines and Meld Lines

Traditionally, the *meeting angle* is used to differentiate weld lines and meld lines. As illustrated in Figure 12.22, a meeting angle, θ , smaller than 135° produces a weld line; greater than 135° , a meld line. Normally, weld lines are considered to be of lower quality than meld lines, since relatively less molecular diffusion occurs across a weld line after it is formed.

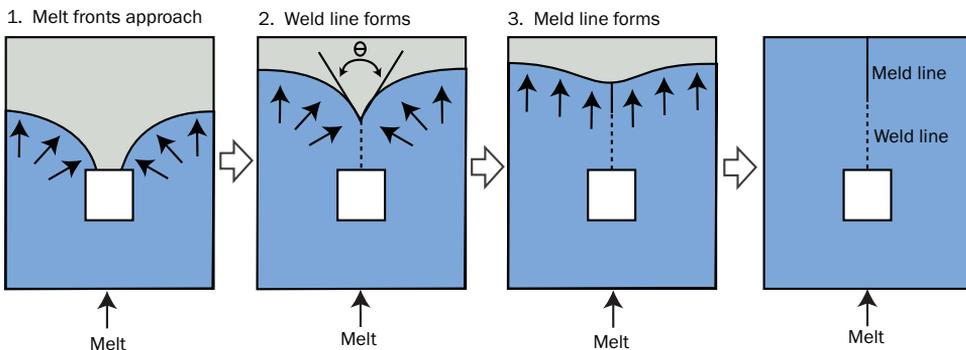


Figure 12.22 Weld and meld lines

12.17.2 Problems Caused by Weld Lines

Weld lines are generally undesirable when part strength and surface appearance are major concerns. This is especially true with fiber-reinforced materials, because the fibers do not bridge the weld lines and often are oriented parallel to them, as illustrated in Figure 12.23.

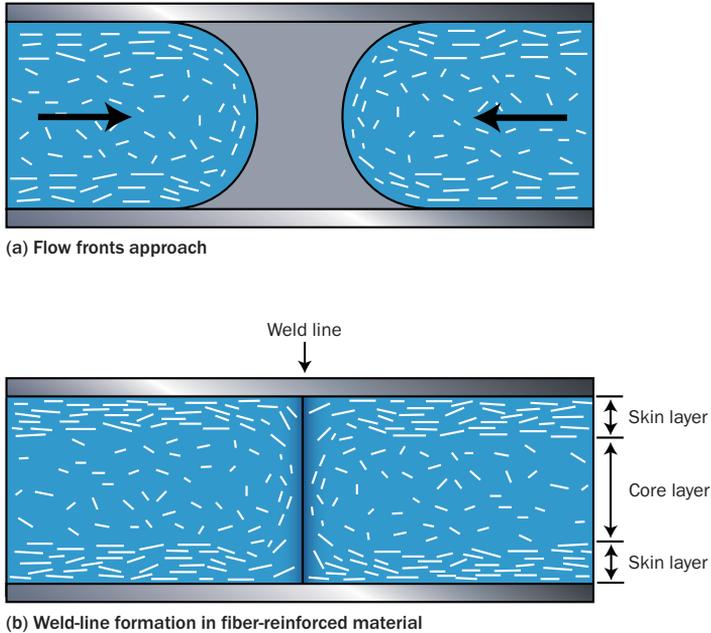


Figure 12.23 Fiber distribution parallel to the weld line leads to a weaker bond

12.17.3 Strength of Weld Lines

The exact strength of the weld line depends on the ability of the flow fronts to weld (or knit) to each other. The strength of the weld-line area can be from 10 to 90% as strong as the pure material used. With such a wide range possible, the conditions that are favorable to better weld line quality are worth examining:

- High injection pressure and speed
- High melt and mold-wall temperature
- Formation of the weld lines closer to the gate
- A temperature difference of less than 10°C between the two emerging flow fronts.

If a weld line forms before the filling is complete and is immediately subject to additional packing pressure, the weld line will typically be less visible and stronger. For complex part geometry, flow simulation helps to predict the weld/meld line position with respect to changes in the tool design and to monitor the temperature difference.

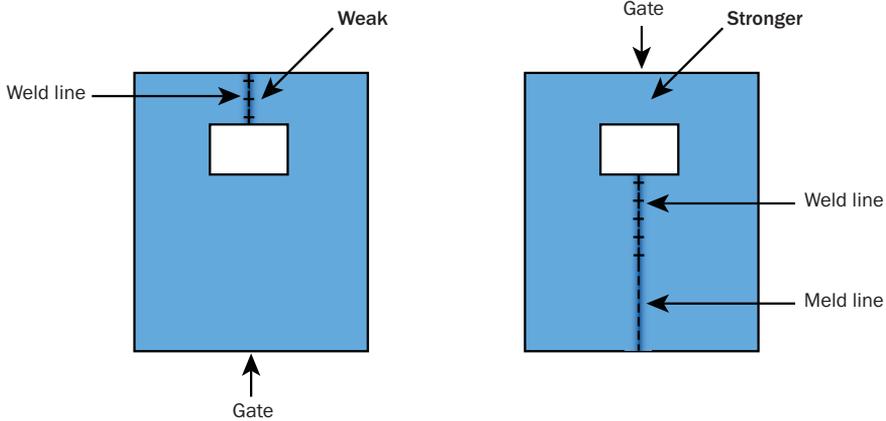


Figure 12.24 Changing the weld-line position by modifying the gate location

12.17.4 Remedies

12.17.4.1 Alter the Part Design

Increase the wall thickness. This will facilitate the transmission of pressure and maintain a higher melt temperature.

Adjust the gate position.

12.17.4.2 Alter the Mold Design

Increase the size of gate and runners.

Place a vent in the area of the weld/meld line. This will eliminate entrapped air, which would further weaken the weld/meld-line.

Change the gate design to eliminate weld/meld lines or to form them closer to the gate at a higher temperature and under a higher packing pressure.

12.17.4.3 Adjust the Molding Conditions

Increase the melt temperature, injection speed, or injection pressure.

Appendix A: Injection Molding

- Injection-molding overview
- Development of the injection-molding machine
- Development of the injection-molding process
- Co-injection (sandwich) molding
- Fusible core injection molding
- Gas-assisted injection molding
- Injection-compression molding
- Lamellar (microlayer) injection molding
- Live-feed injection molding
- Low-pressure injection molding
- Push-pull injection molding
- Reactive molding
- Structural foam injection molding
- Thin-wall molding

A.1 Injection-molding Overview

A.1.1 Process

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

A.1.2 Applications

By weight, approximately 32% of all plastics processed go through injection-molding machines. Historically, the invention of various new alternative processes, such as the reciprocating screw machine, and the application of computer simulation to the design and manufacture of plastics parts are the major milestones of injection molding.

A.2 Development of the Injection-molding Machine

Since its introduction in the early 1870s, the injection-molding machine has undergone significant modifications and improvements. In particular, the invention of the reciprocating screw machine has revolutionized the versatility and productivity of the thermoplastic injection-molding process.

A.2.1 Benefits of the Reciprocating Screw

Apart from obvious improvements in machine control and machine functions, the major development for the injection-molding machine is the change from a plunger mechanism to a reciprocating screw. Although the plunger-type machine is inherently simple, its popularity was limited because of the slow heating rate through pure conduction only. The reciprocating screw, as shown in Figure A.1, can plasticize the material more quickly and uniformly with its rotating motion. In addition, it is able to inject the molten polymer in a forward direction, as a plunger.

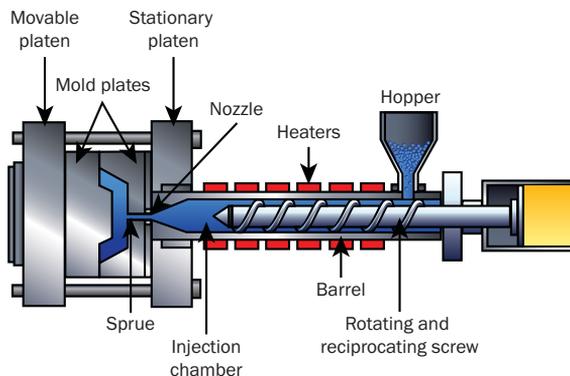


Figure A.1 The reciprocating-screw injection-molding machine

A.3 Development of the Injection-molding Process

The injection-molding process was first used only with thermoplastic polymers. Advances in the understanding of materials, improvements in molding equipment, and the needs of specific industry segments have expanded the use of the process to areas beyond its original scope.

A.4 Alternative Injection-molding Processes

During the past several decades, numerous attempts have been made to develop injection-molding processes to produce parts with special design features and properties. Alternative processes derived from conventional injection molding have created a new era for additional applications, more design freedom, and special structural features. These efforts have resulted in a number of processes, including:

- Co-injection (sandwich) molding (Section A.4.1)
- Fusible core injection molding (Section A.4.2)
- Gas-assisted injection molding (Section A.4.3)
- Injection-compression molding (Section A.4.4)
- Lamellar (microlayer) injection molding (Section A.4.5)
- Live-feed injection molding (Section A.4.6)
- Low-pressure injection molding (Section A.4.7)
- Push-pull injection molding (Section A.4.8)
- Reactive molding (Section A.4.9)
- Structural foam injection molding (Section A.4.10)
- Thin-wall molding (Section A.4.11)

A.4.1 Co-injection (Sandwich) Molding

Co-injection molding involves sequential or concurrent injection of two different but compatible polymer melts into a cavity. The materials laminate and solidify. This process produces parts that have a laminated structure, with the core material embedded between the layers of the skin material. This innovative process offers the inherent flexibility of using the optimal properties of each material or modifying the properties of the molded part.

Figure A.2 shows the steps involved in the co-injection molding process.

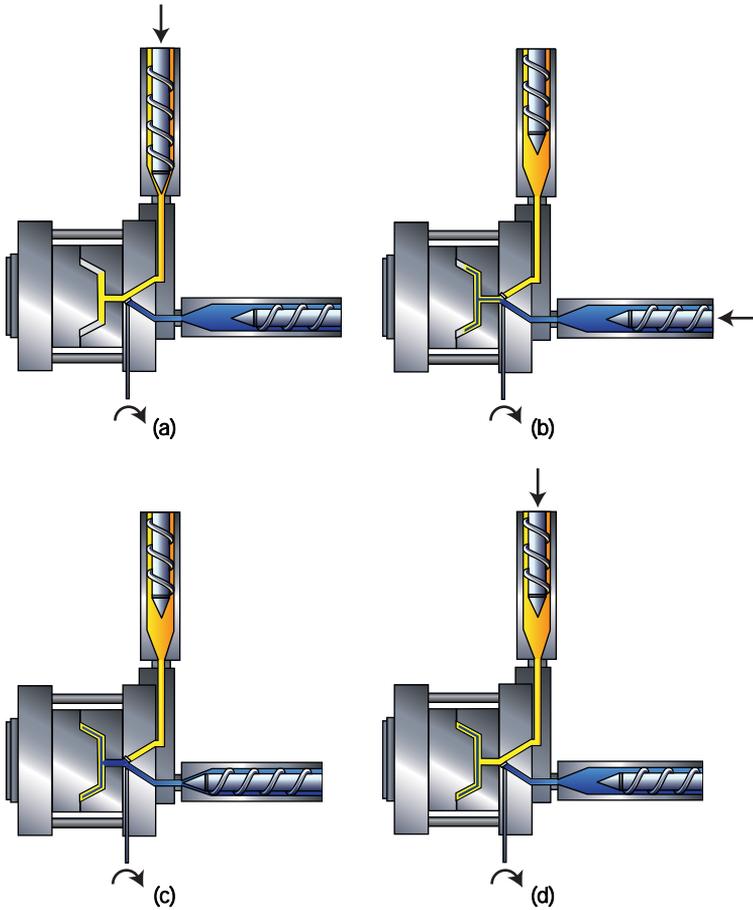


Figure A.2 Four stages of co-injection molding: (a) Short shot of skin polymer melt is injected into the mold. (b) Injection of core polymer melt until cavity is nearly filled, as shown in (c). (d) Skin polymer is injected again to purge the core polymer away from the sprue.

A.4.2 Fusible Core Injection Molding

The fusible (lost, soluble) core injection-molding process illustrated below produces single-piece, hollow parts with complex internal geometry. This process molds a core inside the plastic part. After the molding, the core will be physically melted or chemically dissolved, leaving its outer geometry as the internal shape of the plastic part.

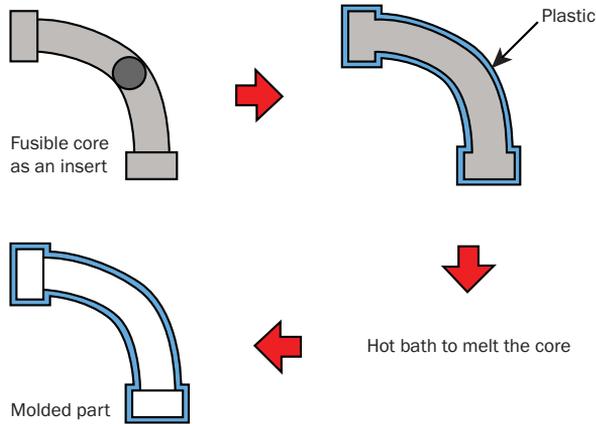


Figure A.3 Fusible (lost, soluble) core injection molding

A.4.3 Gas-assisted Injection Molding

The gas-assisted injection molding process begins with a partial or full injection of polymer melt into the mold cavity. Compressed gas is then injected into the core of the polymer melt to help fill and pack the mold. This process is illustrated in Figure A.4.

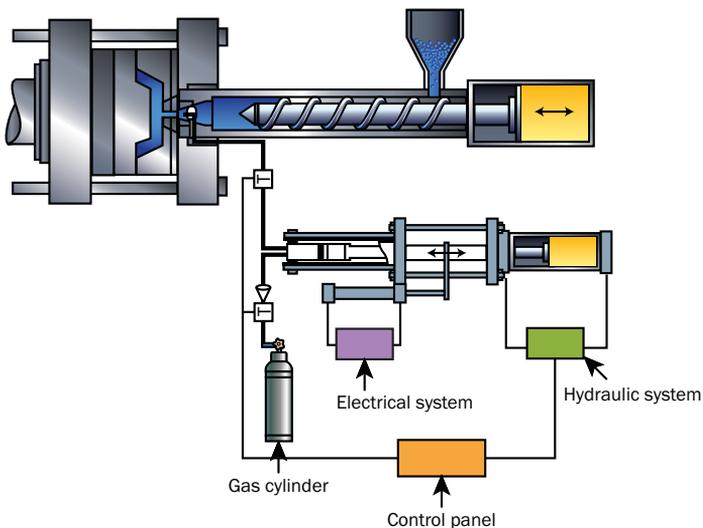


Figure A.4 Gas-assisted injection molding

A.4.3.1 Benefits of the Gas-assist Process

The gas-assisted injection-molding process is capable of producing hollow, light-weight, rigid parts that are free of sink marks and less likely to warp. Other advantages include:

- Reduced cycle time
- Reduced pressure and clamp force tonnage
- Part consolidation with both thick and thin sections

A.4.3.2 Typical Applications

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

- Tube-and rod-like parts, where the process is used primarily for saving material, reducing the cycle time by coring out the part, and incorporating the hollowed section with product function. Examples are clothes hangers, grab handles, chair armrests, shower heads, and water faucet spouts.
- Large, sheet-like, structural parts with a built-in gas-channel network, where the process is used primarily for reducing part warpage and clamp tonnage as well as to enhance rigidity and surface quality. Examples are automotive panels, business machine housings, outdoor furniture, and satellite dishes.
- Complex parts consisting of both thin and thick sections, where the process is used primarily for decreasing manufacturing cost by consolidating several assembled parts into one single design. Examples are television cabinets, computer printer housing bezels, and automotive parts.

A.4.4 Injection-compression Molding

The injection-compression molding process is an extension of conventional injection molding. After a pre-set amount of polymer melt is fed into an open cavity, it is compressed, as shown in Figure A.5. The compression can also take place when the polymer is to be injected. The primary advantage of this process is the ability to produce dimensionally stable, relatively stress-free parts at a low clamp tonnage (typically 20 to 50% lower).

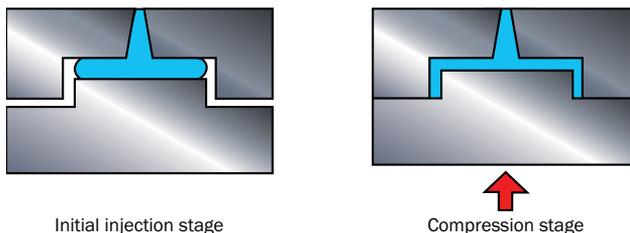


Figure A.5 Injection-compression molding

A.4.5 Lamellar (Microlayer) Injection Molding

This process uses a feedblock and layer multipliers to combine melt streams from dual injection cylinders. It produces parts from multiple resins in distinct microlayers, as shown in Figure A.6. Combining different resins in a layered structure enhances a number of properties, such as the gas barrier property, dimensional stability, heat resistance, and optical clarity.

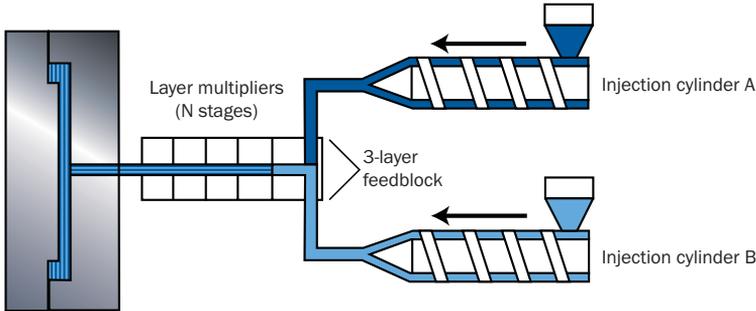


Figure A.6 Lamellar (microlayer) injection molding

A.4.6 Live-feed Injection Molding

The live-feed injection molding process applies oscillating pressure at multiple polymer entrances to cause the melt to oscillate, as shown in Figure A.7. The action of the pistons keeps the material in the gates molten while different layers of molecular or fiber orientation are being built up in the mold from solidification. This process provides a means of making simple or complex parts free from voids, cracks, sink marks, and weld-line defects.

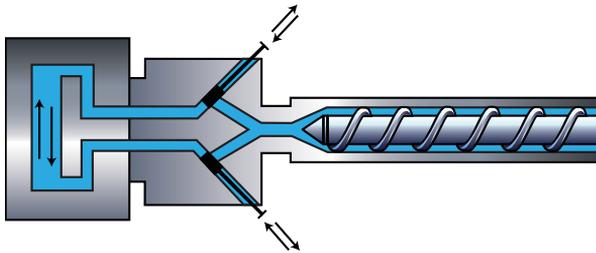


Figure A.7 Live-feed injection molding

A.4.7 Low-pressure Injection Molding

Low-pressure injection molding is essentially an optimized extension of conventional injection molding (see Figure A.8). Low pressure can be achieved by properly programming the screw revolutions per minute, hydraulic back pressure, and screw speed to control the melt temperature and the injection speed. It also makes use of a generous gate size or a number of valve gates that open and close sequentially to reduce the flow length. The packing stage is eliminated with a generally slow and controlled injection speed. The benefits of low-pressure injection molding include a reduction of the clamp force tonnage requirement, less costly molds and presses, and lower stress in the molded parts.

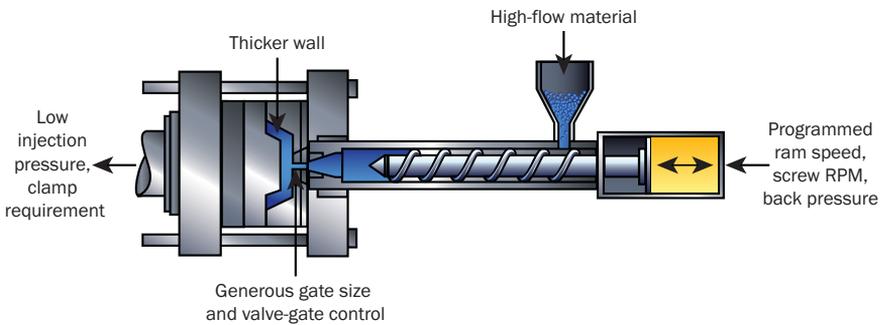


Figure A.8 Low-pressure injection molding

A.4.8 Push-pull Injection Molding

The push-pull injection molding process uses a conventional twin-component injection system and a two-gate mold to force material to flow back and forth between a master injection unit and a secondary injection unit, as shown in Figure A.9. This process eliminates weld lines, voids, and cracks, and controls the fiber orientation.

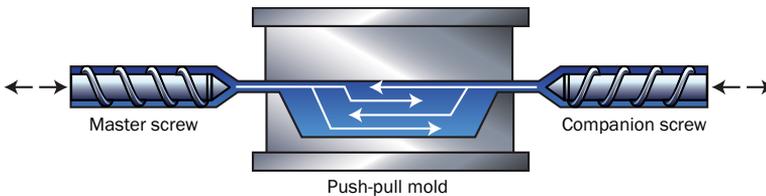


Figure A.9 Push-pull injection molding

A.4.9 Reactive Molding

Unlike thermoplastics, reactive materials undergo simultaneous forming and polymerization during the molding process. The cross-linked polymer structure generally imparts improved mechanical properties and greater heat and environmental resistance.

Table A.1: Types of reactive materials

Acrylic	Phenolics
Alkyds	Polyurethane polyols
Allyl diglycol carbonate	Polyurethane isocyanates
DAIP	Polyurethane systems
DAP	Silicone
Epoxy	Silicone/polymide
Fluorosiliconeurea	Urea
Melamine	Unsaturated polyesters
Melamine/phenolic	

A.4.9.1 Processing

Major reactive molding processes include reactive injection molding (RIM) and composites processing, such as resin transfer molding (RTM) and structural reactive injection molding (SRIM). The typically low viscosity of the reactive materials permits large and complex parts to be molded with relatively lower pressure and clamp tonnage than required for thermoplastics molding. Reactive resins can also be used in the composite processes. For example, to make high-strength and low-volume large parts, RTM and SRIM can be used to include a preform made of long fibers. The encapsulation of microelectronic IC chips is another area receiving more attention than ever before.

The adaptation of injection molding to these materials includes only a small increase in the feed mechanism (barrel) temperature to avoid pre-curing. The cavity, however, is usually hot enough to initiate chemical cross-linking. As the warm pre-polymer is forced into the cavity, heat is added from the cavity wall from both the viscous (frictional) heating of the flow and the heat released by the reacting components. The temperature of the part often exceeds the temperature of the mold. When the reaction is sufficiently advanced for the part to be rigid (even at a high temperature) the cycle is complete and the part is ejected.

A.4.9.2 Design Considerations

The mold and process design for injection molding of reactive materials is much more complex because of the chemical reaction that takes place during the filling and packing stages. For instance, slow filling often causes premature gelling and a resultant short shot, while fast filling could induce the turbulent flow that creates internal porosity. Improper

control of mold-wall temperature and/or inadequate part thickness will cause either moldability problems during injection or scorching of the materials. Computer simulation is generally recognized as a more cost-effective tool than the conventional and time-consuming trial-and-error method for tool and process debugging.

A.4.10 Structural Foam Injection Molding

Structural foam molding produces parts consisting of solid external skin surfaces surrounding an inner cellular (or foam) core, as illustrated in Figure A.10. This process is suitable for large, thick parts subject to bending loads in their end-use application. Structural foam parts can be produced with both low and high pressure, using nitrogen gas or chemical blowing agents.

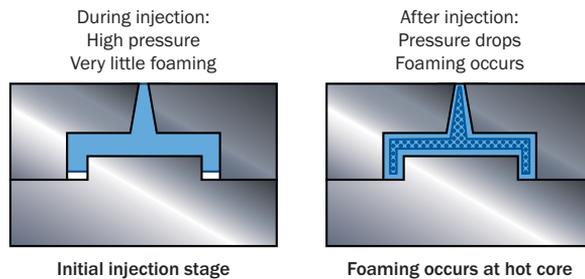


Figure A.10 Structural foam injection molding

A.4.11 Thin-wall Molding

The term *thin-wall* is relative. Conventional plastic parts are typically 2 to 4 mm thick. Thin-wall designs are called *advanced* when thicknesses range from 1.2 to 2 mm and *leading-edge* when the dimension is below 1.2 mm. Another definition of thin-wall molding is based on the flow-length-to-wall-thickness ratios. Typical ratios for these thin-wall applications range from 100:1 to 150:1 or more.

A.4.11.1 Typical Applications

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

A.4.11.2 Processing

Because thin-wall parts freeze off quickly, they require high melt temperatures, high injection speeds, and very high injection pressures if multiple gates or sequential valve gating are not used. An optimized ram-speed profile helps to reduce the pressure requirement.

Due to the high velocity and shear rate in thin-wall molding, orientation occurs more readily. To help minimize anisotropic shrinkage in thin-wall parts, it is important to pack the part adequately while the core is still molten.

Molders should watch for excessive residence time, melt temperatures, or shear—all of which can cause material degradation.

A.4.11.3 Design

We recommend designing parts with styling lines and curved surfaces to boost stiffness and enhance part aesthetics. Impact strategies involve using unreinforced plastic housing to absorb the load or using filled thermoplastics to transfer it. For either case, you'll need to fasten internal components snugly and avoid stress concentration and sharp notches.

Large gates, greater than the wall thickness, are generally used to ensure sufficient material flow during packing.

A.4.11.4 Computer Simulation

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

- Pressure dependence of viscosity, to account for melt viscosity increases with increasing pressure
- Spatial variation of the density (PVT or fast-cooling PVT), to account for the pressure and temperature dependence of density or the effect of fast cooling rate
- Compression work, to account for the additional heating due to the compression work

Appendix B: Injection-molding Machine: System and Operations

- Injection-molding machine
- Machine components
- Molded system
- Machine operating sequence
- Screw operation
- Secondary operations

B.1 Injection-molding Machine

B.1.1 Components

For thermoplastics, the injection-molding machine converts granular or pelleted raw plastic into final molded parts via a melt, inject, pack, and cool cycle. A typical injection molding machine consists of the following major components, as illustrated in Figure A.1.

- Injection system
- Hydraulic system
- Mold system
- Clamping system
- Control system

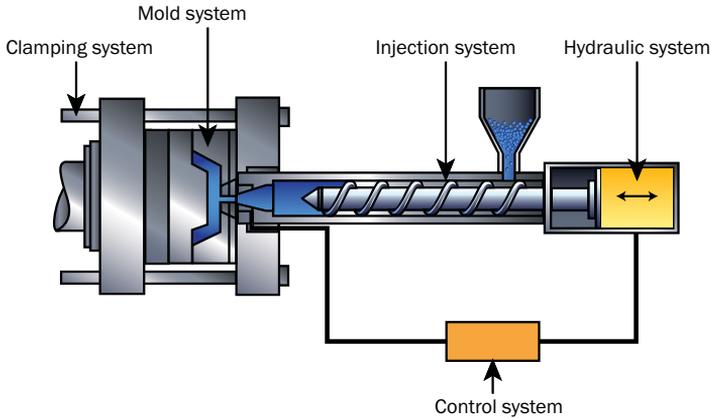


Figure A.1 A single screw injection-molding machine for thermoplastics

B.1.2 Machine Specification

Clamp tonnage and shot size are commonly used to quickly identify the size of the injection-molding machine for thermoplastics. Other parameters include injection rate, injection pressure, screw design, mold thickness, and the distance between tie bars.

B.1.3 Machine Function

Injection-molding machines can be generally classified into three categories, based on machine function:

- General-purpose machines
- Precision, tight-tolerance machines
- High-speed, thin-wall machines

B.1.4 Auxiliary Equipment

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

B.2 Machine Components

B.2.1 Injection System

The injection system consists of a hopper, a reciprocating screw and barrel assembly, and an injection nozzle, as shown in Figure A.2. This system confines and transports the plastic as it progresses through the feeding, compressing, degassing, melting, injection, and packing stages.

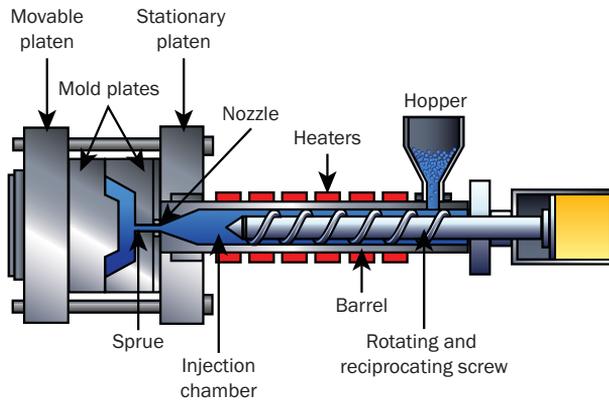


Figure A.2 A single screw injection-molding machine for thermoplastics, showing the plasticizing screw, a barrel, band heaters to heat the barrel, a stationary platen, and a movable platen

B.2.1.1 The Hopper

Thermoplastic material is supplied to molders in the form of small pellets. The hopper on the injection molding machine holds these pellets. The pellets are gravity-fed from the hopper through the hopper throat into the barrel and screw assembly.

B.2.1.2 The Barrel

As shown in Figure A.2, the barrel of the injection molding machine supports the reciprocating plasticizing screw. It is heated by the electric heater bands.

B.2.1.3 The Reciprocating Screw

The reciprocating screw is used to compress, melt, and convey the material. The reciprocating screw consists of three zones, illustrated in Figure A.3:

- Feeding zone
- Transition (or compressing) zone
- Metering zone

While the outside diameter of the screw remains constant, the depth of the flights on the reciprocating screw decreases from the feed zone to the beginning of the metering zone. These flights compress the material against the inside diameter of the barrel, which creates viscous (shear) heat. This shear heat is mainly responsible for melting the material. The heater bands outside the barrel help maintain the material in the molten state. Typically, a molding machine can have three or more heater bands or zones with different temperature settings.

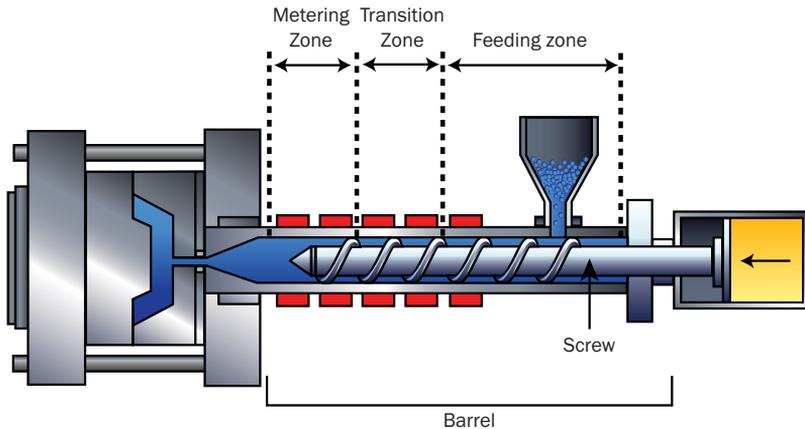


Figure A.3 A reciprocating screw, showing the feeding zone, transition (or compressing) zone, and metering zone

B.2.1.4 The Nozzle

The nozzle connects the barrel to the sprue bushing of the mold and forms a seal between the barrel and the mold. The temperature of the nozzle should be set to the material's melt temperature or just below it, depending on the recommendation of the material supplier. When the barrel is in its full forward processing position, the radius of the nozzle should nest and seal in the concave radius in the sprue bushing with a locating ring, as shown in Figure A.4 (a). During purging of the barrel, the barrel backs out from the sprue, as shown in Figure A.4 (b), so the purging compound can free fall from the nozzle.

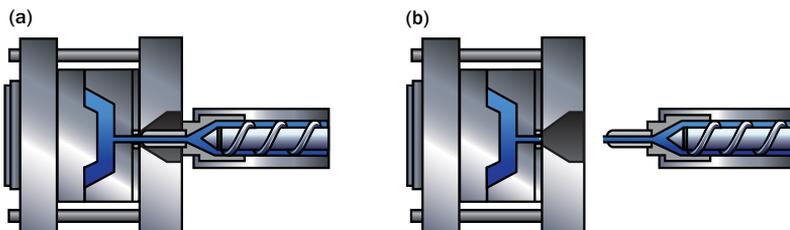


Figure A.4 (a) Nozzle with barrel in processing position (b) Nozzle with barrel backed out for purging

B.2.2 Mold System

The mold system consists of tie bars, stationary and moving platens, in addition to molding plates (bases), which house the cavity, sprue and runner systems, ejector pins, and cooling channels, as shown in Figure A.5. The mold is essentially a heat exchanger in which the molten thermoplastic solidifies to the desired shape and dimensional details defined by the cavity.

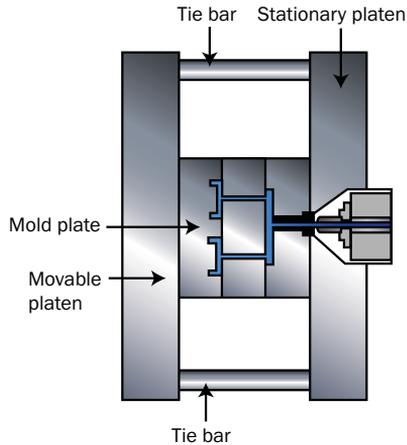


Figure A.5 A typical (three-plate) molding system

A mold system is an assembly of platens and molding plates typically made of tool steel. The mold system shapes the plastics inside the mold cavity (or matrix of cavities) and ejects the molded part(s). The stationary platen is attached to the barrel side of the machine and is connected to the moving platen by the tie bars. The cavity plate is generally mounted on the stationary platen and houses the injection nozzle. The core plate moves with the moving platen guided by the tie bars. Occasionally, the cavity plate is mounted to the moving platen and the core plate and a hydraulic knock-out (ejector) system is mounted to the stationary platen.

B.2.2.1 Two-plate Mold

The vast majority of molds consist essentially of two halves, as shown in Figure A.6 (a). This kind of mold is used for parts that are typically gated on or around their edge, with the runner in the same mold plate as the cavity.

B.2.2.2 Three-plate Mold

The three-plate mold is typically used for parts that are gated away from their edges. The runner is in between two plates, separate from the cavity and core, as shown in Figure A.6 (b).

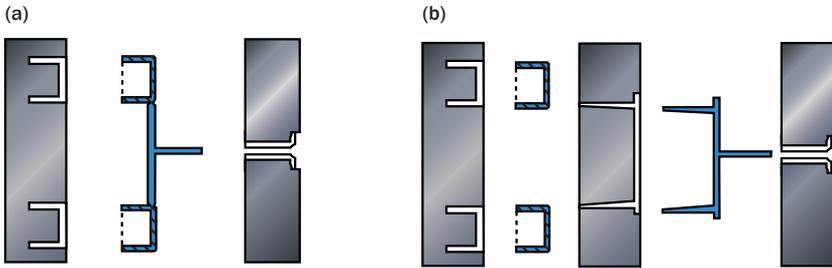


Figure A.6 (a) Two-plate mold (b) Three-plate mold

B.2.2.3 Cooling Channels (Circuits)

Cooling channels are passageways located within the body of a mold, through which a cooling medium (typically water, steam, or oil) circulates. Their function is the regulation of temperature on the mold surface. Cooling channels can also be combined with other temperature control devices, like bafflers, bubblers, and thermal pins or heat pipes.

B.2.3 Hydraulic System

The hydraulic system on the injection-molding machine provides the power to open and close the mold, build and hold the clamp tonnage, turn the reciprocating screw, drive the reciprocating screw, and energize ejector pins and moving mold cores. A number of hydraulic components are required to provide this power, including pumps, valves, hydraulic motors, hydraulic fittings, hydraulic tubing, and hydraulic reservoirs.

B.2.4 Control System

The control system provides consistency and repeatability in machine operation. It monitors and controls the processing parameters, including the temperature, pressure, injection speed, screw speed and position, and hydraulic position. The process control has a direct impact on the final part quality and the economics of the process. Process control systems can range from a simple relay on/off control to an extremely sophisticated microprocessor-based closed-loop control.

B.2.5 Clamping System

The clamping system opens and closes the mold, supports and carries the constituent parts of the mold, and generates sufficient force to prevent the mold from opening. Clamping force

can be generated by a mechanical (toggle) lock, hydraulic lock, or a combination of the two basic types.

B.3 Molded System

A typical molded system consists of the delivery system and the molded part(s), as shown in Figure A.7.

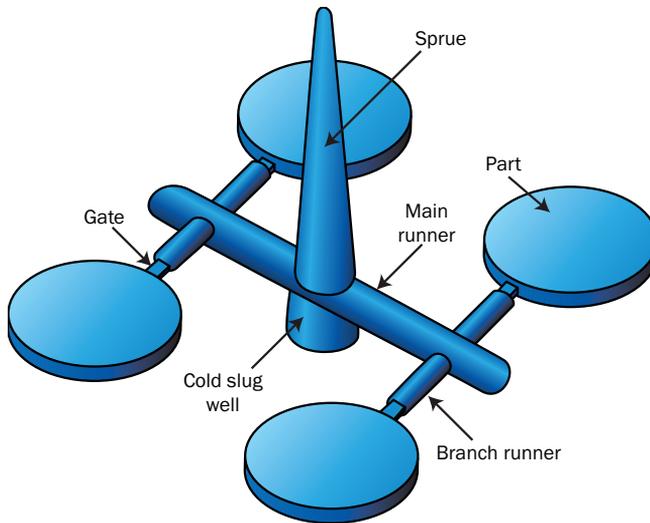


Figure A.7 The molded system includes a delivery system and molded parts.

B.3.1 The Delivery System

The delivery system, which provides passage for the molten plastic from the machine nozzle to the part cavity, generally includes:

- Sprue
- Cold slug wells
- Main runner
- Branch runners
- Gates

The delivery system design has a great influence on the filling pattern and thus the quality of the molded part.

B.3.1.1 Cold Runners

After molding the cold-runner delivery system is trimmed off and recycled. Therefore, the delivery system is normally designed to consume minimum material, while maintaining the function of delivering molten plastic to the cavity in a desirable pattern.

B.3.1.2 Hot Runners

The hot-runner (or runnerless) molding process keeps the runners hot to maintain the plastic in a molten state at all times. Since the hot-runner system is not removed from the mold with the molded part, it saves material and eliminates the secondary trimming process. The volume of the hot runner should be less than the shot size.

B.4 Machine Operating Sequence

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

B.4.1 Process Cycle

The basic injection-molding machine operations are shown in Figure A.8.

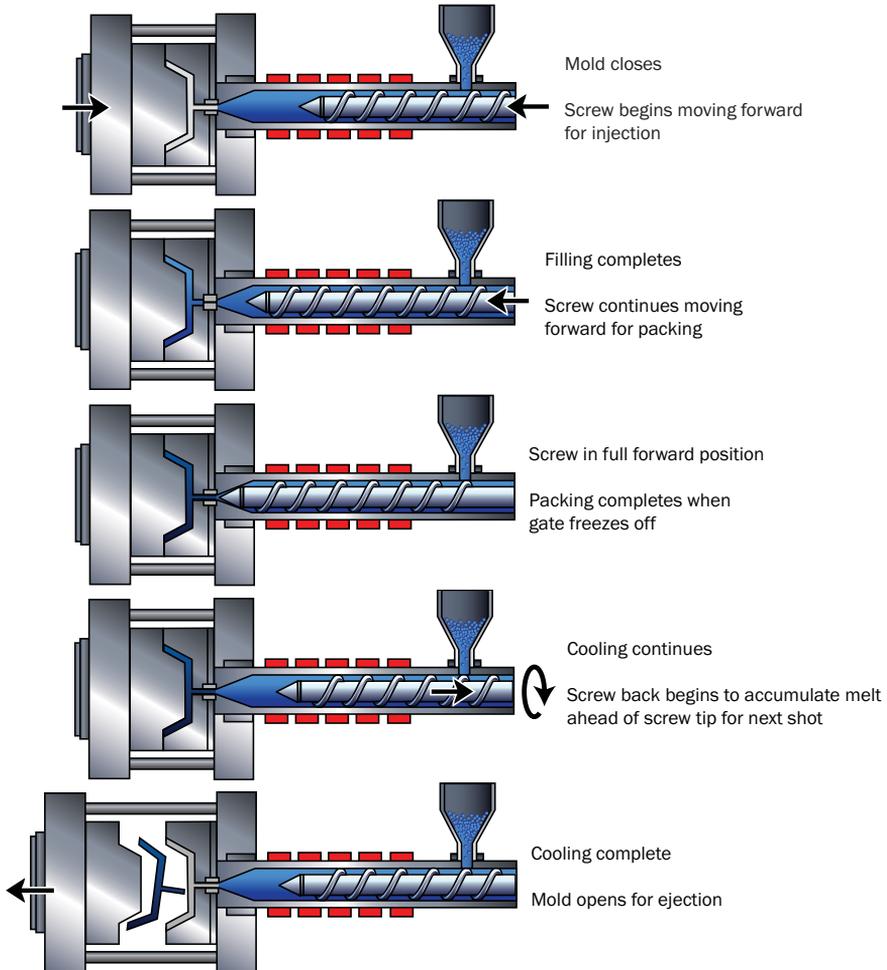


Figure A.8 The basic injection-molding machine operations

B.4.1.1 Cycle Time

Typical process cycle time varies from several seconds to tens of seconds, depending on the part weight, part thickness, material properties, and the machine settings specific to a given process.

B.5 Screw Operation

The reciprocating screw is used to plasticize the plastic pellets using various screw rotation speeds (revolutions per minute, or RPMs), inject the molten plastics as a plunger at various speeds and shot volumes, and control the pressure level in the molten plastic charge in front of the screw. Several of its operations are discussed here.

B.5.1 Back Pressure

Back pressure is the amount of pressure exerted on the material volume ahead of the screw as the screw is pushed back in preparation for the next shot.

B.5.1.1 Setting the Maximum Back Pressure

Typically, all machines have an adjustment for the maximum back pressure. This *screw-back* stage stops when the screw reaches a preset position. The stop position is manually set, based on the amount of material required to fill the mold's cavity and runner system. When the machine is ready to inject the shot, the screw then plunges the material ahead of the screw forward, injecting it into the mold. While the injected material is cooled in the mold after the injection, the screw-back stage is re-initiated and the molding cycle repeated.

B.5.2 Injection Speed

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

B.5.2.1 Setting the Injection Speed

For most engineering resins, the ram speed should be set to the fastest setting that the part design and process will allow for technical and economic reasons. However, slower injection speed at the beginning of injection may be necessary to avoid turbulent flow and jetting, as material passes through the restrictive areas (e.g. gates). The injection speed should be reduced again toward the end of injection to avoid flashing at the end of stroke, and to enhance the formation of homogenous weld lines after a divided flow.

B.5.3 Screw Rotation Speed

The screw rotation speed (RPM) is the rate at which the plasticizing screw rotates. The faster the screw rotates, the faster the material is compressed by the screw flights, increasing the amount of shear heating.

B.5.4 Cushion

The cushion is the difference in the final forward position of the screw and its maximum allowable forward position. If the screw were allowed to travel its full stroke and stop mechanically against the nozzle, the cushion would be zero. Typically a cushion of 3 to 10 mm (1/8 to 3/8 in) is used.

B.6 Secondary Operations

After a part is ejected, the delivery system (sprue, runners, and gates) is trimmed off as a secondary operation. For some applications, additional secondary operations are needed for assembly or decoration. Detailed descriptions of these secondary operation procedures can be found in design handbooks from material suppliers.

B.6.1 Assembly

Secondary operations for assembling parts include:

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

B.6.2 Decoration

Secondary operations for decorating the plastic parts include:

- Applique: a surface covering applied by heat and pressure
- Printing: a process of making a mark or impression onto a substrate for decorative or informational purposes.

B.6.3 Other Secondary Operations

Other secondary operations include:

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

Appendix C: Injection-molding Process Control

- Importance of process conditions
- Setting machine process conditions

C.1 Importance of Process Conditions

The quality of the molded part is greatly influenced by the conditions under which it is processed. See, for example, the process window shown in Figure A.1. As you lower the temperature, higher pressure is needed to deliver the polymer melt into the cavity. If the temperature is too high, you risk causing material degradation. If the injection pressure is too low, a short shot could result. If the pressure is too high, you will flash the mold.

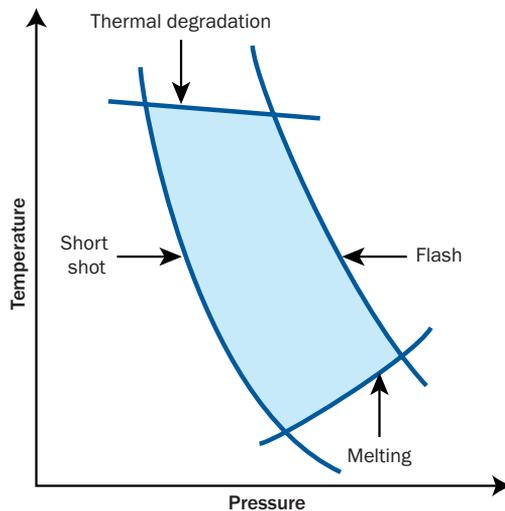


Figure A.1 Process window showing the influence of pressure versus temperature

C.1.1 Setting Machine Process Conditions

Before setting process conditions, you should make certain the molding machine is in proper working order, and that the mold you plan to use was designed for the particular machine you plan to use. Follow the step-by-step procedure provided below to control the settings on your machine.

C.1.1.1 Step 1—Set the Melt Temperature

Melt temperature is one of the most important factors in molding plastic parts. If it is too low, the resin might not be completely melted or it might be too sticky to flow. If the melt temperature is too high, the resin could degrade, especially if the resin is POM or PVC. Suggested melt and mold temperatures for specific materials are available from the resin supplier. Appendix D contains a reference list of resins, their general properties, and typical applications. Appropriate melt and mold temperatures for several materials are listed in Table 2.

Setting Heater Band Temperatures: Most melting of the resin occurs because of the frictional heating from the screw rotation inside the barrel. The barrel heater bands serve mainly to keep the resin at the appropriate temperature. Typically there are three to five temperature zones or heater bands on the cylinder. The rules for setting the heater band temperatures are as follows:

- The last temperature zone, nearest the hopper, should be about 40 to 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.

Air-shot Temperature: The actual melt temperature, or air-shot temperature, is usually higher than the heater band controller setting. This difference is because of the influence of back pressure and screw rotation on frictional heating and the melt temperature, as mentioned above. (You can measure the actual melt temperature by quickly sticking a probe thermometer into an air shot with the nozzle backed away from the mold.)

C.1.1.2 Step 2—Set the Mold Temperature

Suggested melt and mold temperatures for specific materials are available from the resin supplier. Appropriate melt and mold temperatures for many generic, base resins are listed in Table 2. The mold temperature can be measured by using a thermometer. As illustrated in Figure A.2, the average cavity surface temperature will be higher than the temperature of the coolant during production. Thus, you should set the coolant temperature to be 10 to 20°C (18 to 36°F) lower than the required mold temperature. 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.

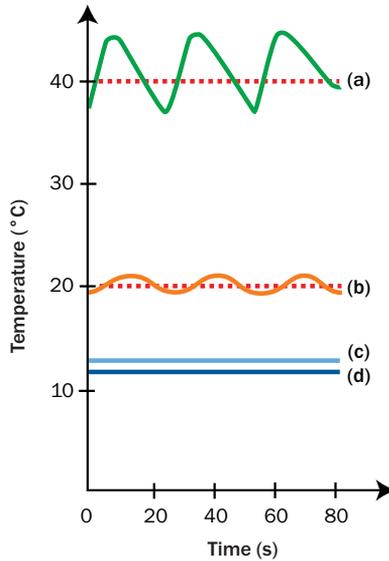


Figure A.2 Temperature-time curve at various locations in the mold (a) Mold cavity surface (b) Cooling channel wall (c) Cooling channel outlet (d) Cooling channel inlet

Use the lowest temperature setting to achieve the shortest cycle time. However, you might try using higher temperatures to improve the appearance of the part. A higher mold temperature produces a higher gloss and more crystallization.

Considering Temperature Difference: For parts with a deep core, a lower coolant temperature is needed for the core (moving plate) in order to minimize the temperature difference between the mold surfaces on the core and cavity. A lower surface temperature difference will produce parts with higher quality, at a lower cost. By a rule of thumb, the coolant temperature for fixed and moving plates should not differ by more than 20°C (36°F). This is related to thermal expansion, which can be determined only by the user. A large temperature difference results in differential mold plate thermal expansion, which may cause alignment problems in guide pins, especially in large molds. The mold will sometimes lock up for this reason. The cycle time can be increased to reduce the required coolant temperature difference.

C.1.1.3 Step 3—Set the Switch-over Position

The switch-over position is the ram position where the velocity controlled filling (injection) stage switches to the pressure controlled packing phase. Once the ram is under pressure control, the ram continues to move forward to pressurize the cavity and to compensate for shrinkage. The cushion is the distance from the ram position at the end of the packing phase to the farthest position that the end of the screw can reach, as shown in Figure A.3. The typical cushion distance is about 3 to 10 mm (1/8 to 3/8 in).

pressure to nearly the machine maximum. The purpose is to completely exploit the injection velocity of the machine, so that the pressure setting valve does not limit the velocity. Because the switch-over to packing pressure occurs before the mold is completely filled, no damage will be done to the mold.

C.1.1.7 Step 7—Set the Packing Pressure at 0 Mpa

For now, set the packing pressure at 0 MPa, so the screw will stop when it reaches the switch-over position. This will prevent mold or press damage. In Step 17, the packing pressure is increased to its final setting.

C.1.1.8 Step 8—Set the Injection Velocity to the Machine Maximum

With the highest possible injection velocity within shear rate limits, you can expect less flow resistance, longer flow length, and improved strength in weld lines. However, you may need to create additional vents once you do this.

Proper Venting Minimizes Defects: Insufficient venting causes compression of air trapped in the cavity. This results in very high temperatures and pressures in the cavity, causing burn marks, material degradation, and short shots. You should design a venting system to avoid or minimize the defects caused by trapped air in the mold. Moldflow shows you where weld lines, meld lines, and air trap locations will occur: use these predictions to improve your design. Remember that it is necessary to clean the mold surface and venting system regularly, especially for PVC or ABS/PVC materials.

C.1.1.9 Step 9—Set the Packing Time

The ideal packing time setting is the gate freezing (sealing) time or the part freezing time, whichever is shorter. The gate and part freezing times can be calculated or estimated. The calculated values for the packing time are based on packing analysis results when the frozen layer fraction is 1.0 for the gate. Without packing analysis results, the packing time is estimated to be 10 times the filling time.

C.1.1.10 Step 10—Set Ample Cooling Time

Cooling time can be calculated or estimated. The cooling time is after packing time, as shown in Figure A.4. During cooling the part continues to solidify so it can be ejected, and material for the next shot is prepared. The calculated value of cooling time is from a cooling or packing analysis. Without Moldflow results, the cooling time can be 10 times the filling time. For example, if the predicted filling time is 0.85 seconds, the initial cooling time would be 8.5 seconds. The combination of packing time (if estimated would be 8.5 seconds) and the cooling time should be a high estimate to ensure the part and runner system will be sufficiently solid for ejection.

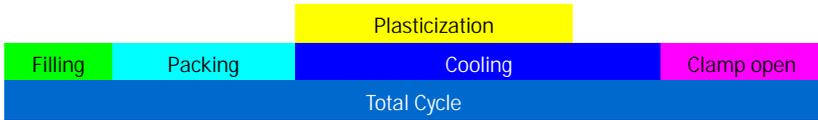


Figure A.4 Cycle time and its components

C.1.1.11 Step 11—Set the Mold Opening Stroke

The mold opening stroke is comprised of the core height, the part height, and the capsize space, as shown in Figure A.5. You should minimize the mold opening stroke. The mold opening speed should be slow at the very beginning, then accelerate, then slow down again at the end of the stroke. The sequence of the mold closing speed is similar to the mold opening speed: slow, fast, slow.

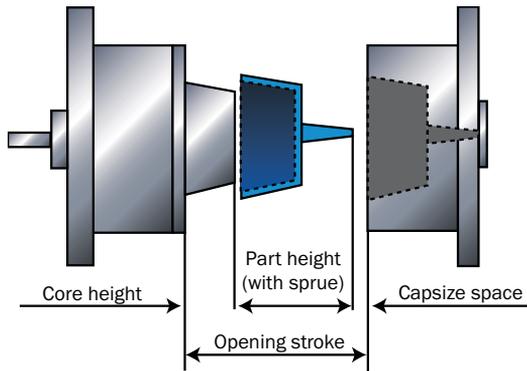


Figure A.5 Required mold opening

C.1.1.12 Step 12—Set the Ejector Stroke, Start Position, and Velocity

Relieve any slides first. The ejector travel should be, at a maximum, the core height. If the machine is equipped with a hydraulic ejector, set the start position at the point where the part is clear of stationary mold parts. (When the ejector velocity is equal to the opening speed, the part remains where it was in relation to the stationary mold part.)

C.1.1.13 Step 13—Set the Mold Open Time

The mold open time is usually set at 2 to 5 seconds. This includes mold opening, ejection of parts from the mold, then mold closing, as shown in Figure A.4. The cycle time is the sum of the filling time, cooling time, and mold open time.

C.1.1.14 Step 14—Mold a Short-shot Series by Increasing Injection Volume

Moldflow provides the part weight and sprue/runner/gate weight. From this information, along with the screw diameter or barrel inner diameter, the total injection volume and the feeding position (see Figure A.3) can be estimated for each shot.

For now, fill only two-thirds of the mold. The packing pressure should already be set at 0 MPa, so that mold filling stops when the screw reaches the switch-over position, thus protecting the mold structure and the press. Next, increase the volume in increments of 5 to 10%, up to 95% of mold filling.

In order to prevent material from escaping from the open nozzle, relieve the back pressure created during plasticizing by drawing back the screw a few millimeters, immediately after the rotation has stopped.

C.1.1.15 Step 15—Switch to Automatic Operation

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

C.1.1.16 Step 16—Set the Injection Volume to 99% Mold Filled

When the process has stabilized (when the same parts are produced each time), adjust the switch-over position to 99% of filling. This will exploit the maximum injection speed in as large a part of the injection as possible.

C.1.1.17 Step 17—Increase the Packing Pressure in Steps

Increase the packing pressure in steps of approximately 10 MPa in the melt. If the first step does not fill the mold completely, increase the injection volume.

De-mold and remove the part. Write the packing pressure on it. This packing pressure series forms a good basis for a more thorough examination. You can then discuss the possibilities and limitations with the customer.

Choose the lowest acceptable packing pressure, as this minimizes the internal stresses in the part and saves material, as well as operating costs. A high packing pressure can cause excessive residual stresses that could warp the part. Molded-in residual stresses can be released somewhat by annealing at around 10°C (18°F) below the heat deflection temperature.

If the material cushion is completely used (see Figure A.3), the last part of the packing pressure time will not be effective. This calls for a change in the injection stroke position, in order to increase the injection volume.

Calculating Injection Pressure: The hydraulic pressure in the injection cylinder can be read on the machine manometer. However, the injection pressure in front of the screw is more important. To calculate the injection pressure you will need to multiply the Intensification ratio (hydraulic pressure by the resin/hydraulic pressure ratio). This ratio is usually found on the molding machine near the injection unit or in the instruction manual for the machine. The ratio is usually in the range of 7 to 15, as shown in Figure A.6 below.

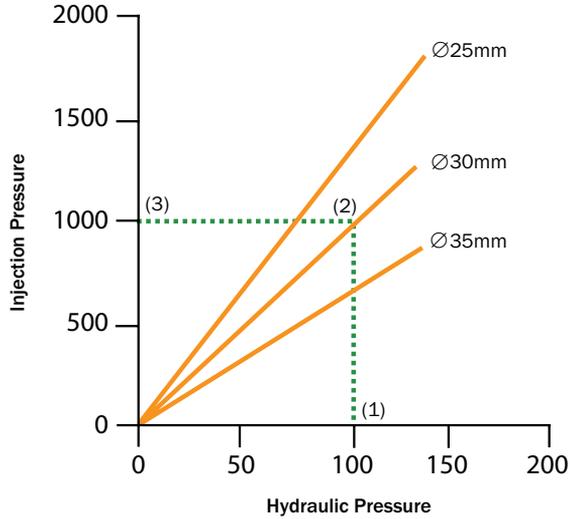


Figure A.6 Intensification ratio for a $\varnothing 30$ mm screw is 11.1

C.1.1.18 Step 18—Minimize the Packing Time

If consistent part dimensions are essential, the following method of packing time determination is more accurate. The gate seal time must be determined. The gate seal time can be determined experimentally on the molding machine with a gate seal study. This involves initially starting with a long packing time and reducing the pack time until the part weights begin to change. This is an indication that the gate is open. When the packing time is decreased, the cooling time should increase the same amount to maintain the same cycle time. For example, Figure A.7 shows that the packing pressure does not influence the part weight after 9 seconds. This is your minimum packing time.

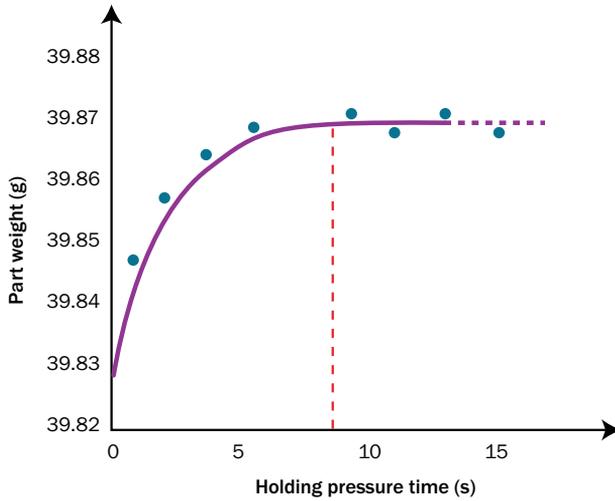


Figure A.7 Determination of the gate/part freezing time by weighing parts manufactured at various packing times

C.1.1.19 Step 19—Minimize the Remaining Cooling Time

Reduce the remaining cooling time until the maximum surface temperature of the part reaches the heat deflection temperature of the material. The heat deflection temperature can be provided by the resin supplier.

Appendix D: Plastic Materials

- What are plastics?
- Classification of plastics
- Thermoplastics
- Thermosets
- Properties and applications of thermoplastics
- Additives, fillers, and reinforcements

D.1 What Are Plastics?

D.1.1 Polymerization Process

Plastics are one group of polymers built from relatively simple units called monomers (or mers) through a chemical polymerization process. This process is illustrated in Figure A.8. Processing polymers into end products mainly involves physical phase change, such as melting and solidification (for thermoplastics) or a chemical reaction (for thermosets).

D.1.2 Structure of Polymers

The basic structure of a polymer molecule can be visualized as a long chain of repeating units, with additional chemical groups forming pendant branches along the primary “backbone” of the molecule, as shown in Figure A.8. Although the term *plastics* has been loosely used as a synonym for polymer and resin, plastics generally represent polymeric compounds that are formulated with plasticizers, stabilizers, fillers, and other additives for purposes of processibility and performance. Other polymeric systems include rubbers, fibers, adhesives, and surface coatings. A variety of processes have been employed to produce the final plastic parts, as illustrated in Figure A.9.

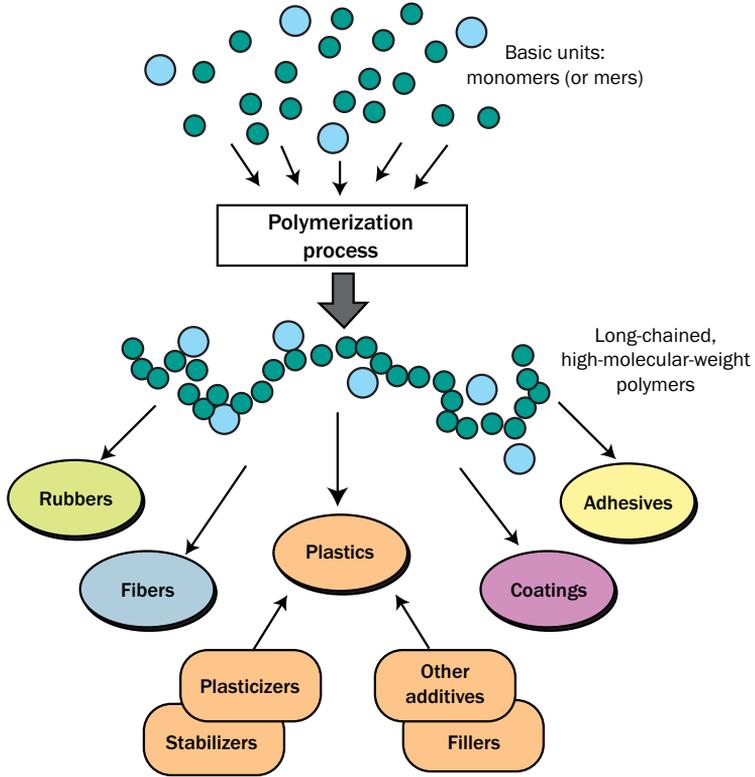


Figure A.8 Polymer family: the formation of plastics and the polymerization process

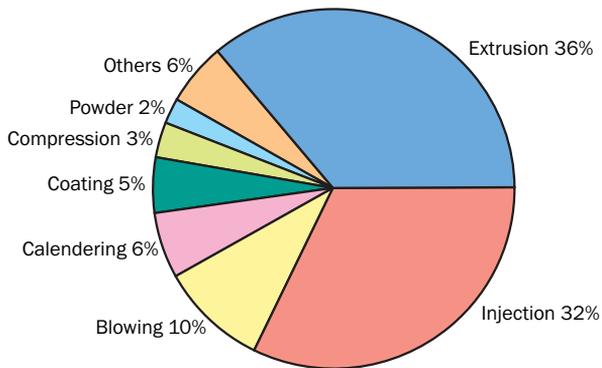


Figure A.9 Plastics consumption by process

D.1.2.1 Structure-dependent Properties

The structural arrangement, size, and chemical constitution of the polymer molecule have a direct influence on its physical and chemical properties. In addition, the macromolecular nature of plastics implies that their material properties may also be dependent on the mechanical and thermal history that the materials experience during processing. For example, the viscosity (which indicates the material's resistance to flow) of a polymer melt increases with increasing molecular weight, but decreases as temperature increases. Further, the aligned molecular orientation that results from strong shear exerted on the material also reduces the viscosity of the polymer melt.

The physical and mechanical properties, as well as the cost of polymers, can be modified by blending a number of polymers or by compounding them with other materials or reinforcing agents. These processes have resulted in the following polymeric systems.

D.1.3 Polymer Alloys and Blends

Polymer alloys and blends are mixed systems of two or more finished polymers. When the combination of polymers has a single glass transition temperature and yields a synergistic effect (i.e., the properties of the mix are better than either of the individual components), the resulting system is termed a polymer alloy. When the resulting product has multiple glass transition temperatures and its properties are the average of the individual components, the material system is referred to as a polymer blend. One of the earliest commercially successful blends was ABS, which combines the chemical resistance, toughness, and rigidity of its components.

D.1.4 Polymer Composites

Polymer composites are materials that incorporate certain reinforcing agents into a polymer matrix to add desirable properties. Low aspect ratio materials—such as single crystal/whisker, and flake-type fillers of clay, talc, and mica—impart increased stiffness. On the other hand, larger aspect ratio reinforcements—such as fibers or filaments of glass, carbon-graphite, aramid/organic, and boron—substantially raise both the tensile strength and the stiffness.

D.2 Classification of Plastics

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

D.2.1 Classes of Plastics

In addition to the broad categories of thermoplastics and thermosets, thermoplastics can be further categorized into amorphous, (semi-)crystalline, or liquid crystal polymers (LCPs), depending on the polymer chain conformation or morphology. The microstructures of these plastics and the effects of heating and cooling on the microstructures are shown in Figure A.10. Other classes include elastomers, copolymers, compounds, commodity resins, and engineering resins. Additives, fillers, and reinforcements are other classifications that relate directly to plastics properties and performance.

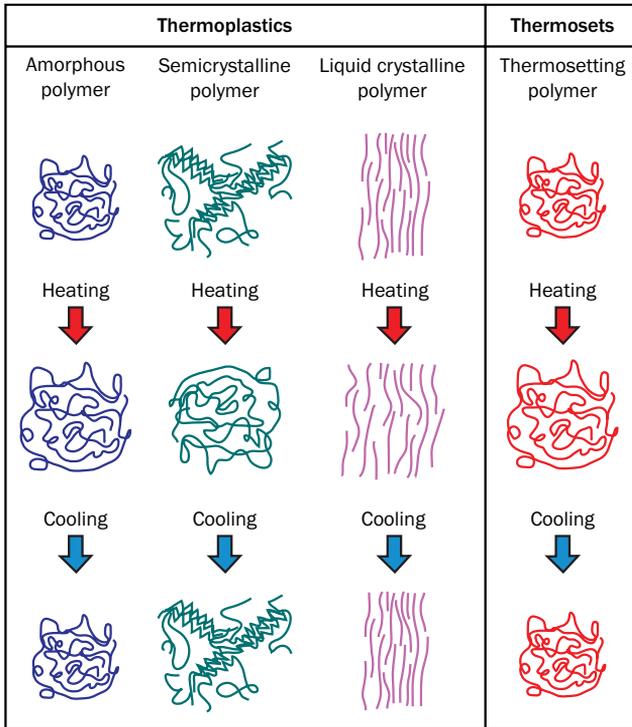


Figure A.10 Microstructure of various plastics and effect of heating and cooling during processing

D.2.2 Structures and Properties of Plastics

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

Table A.1: Structures and properties of thermoplastics and thermosets

	Thermoplastics	Thermosets
Microstructure	Linear or branch molecules No chemical bonds among the molecules	Cross-linking network with chemical bonds among molecules after the chemical reaction
Reaction to heat	Can be re-softened (physical phase change)	Cannot be re-softened after cross-linking without degradation
General properties	Higher impact strength Easier processing Better adaptability to complex shapes	Greater mechanical strength Greater dimensional stability Better heat and moisture resistance.

D.3 Thermoplastics

Thermoplastics typically have high molecular weights resulting from a high degree of polymerization. The long molecular chain, either linear or branched, has side chains or groups that are not attached to other polymer molecules. As a result, thermoplastics can be repeatedly softened (or hardened) by an increase (or decrease) in temperature. This type of phase change without a chemical reaction permits the recycling of thermoplastic scraps, such as the trimmed-off runners and sprues from injection molding. An analogy is the phase change of ice turning into water under heat, and then becoming a solid again when cooled. Although thermoplastics are recyclable, it is very likely that a small degree of chemical change (e.g., oxidation or thermal degradation) takes place during processing, and therefore the properties of recycled polymers may not be equivalent to those of the virgin polymer.

D.3.1 Market Share Distribution of Thermoplastics

Thermoplastics account for more than 70% of all polymers produced. Thermoplastic materials are purchased as pellets or granules. They are melted by heat under pressure into a relatively viscous fluid and shaped into a desirable product or form by cooling. Thermoplastics generally offer higher impact strength, easier processing, and better adaptability to complex designs than do thermosets.

D.3.1.1 Commodity Resins

Among thermoplastics, the commodity resins—for example, high-density polyethylene (HDPE), low-density polyethylene (LDPE), polypropylene (PP), polystyrene (PS), and polyvinyl chloride (PVC)—account for more than 90% of all thermoplastics.

D.3.1.2 Engineering Resins

On the other hand, the engineering resins—such as acetal, acrylonitrile butadiene styrene (ABS), nylon, and polycarbonate (PC)—offer improved performance, including higher mechanical properties, better heat resistance, and higher impact strength. Thus, they demand a higher price.

D.3.2 Structures and Properties of Thermoplastics

Table 1 lists a summary of the relevant structures and properties of amorphous polymers and crystalline polymers.

Table 1: Structures and properties of amorphous and crystalline polymers

	Amorphous Polymers	Crystalline Polymers
Common materials	Acrylonitrile butadiene styrene (ABS) Acrylics (e.g., PAN, PMMA) Polycarbonate (PC) Polystyrene (PS) Polyvinyl chloride (PVC) Styrene acrylonitrile (SAN)	Acetals Nylon Polyethylene (PE) Polypropylene (PP) Thermoplastic Polyesters (e.g., PBT, PET)
Microstructure	Random molecular orientation in both molten and solid phases.	Random molecular orientation in molten phase, but densely packed crystallites occurs in solid phase.
Reaction to heat	Softens over a range of temperatures (no apparent melting temperature)	Fairly distinct melting temperature.
General properties	Transparent Poor chemical resistance Low volumetric shrinkage in molding Generally low strength Generally high melt viscosity Lower heat content	Translucent or opaque Excellent chemical resistance High volumetric shrinkage in molding Generally high strength Generally low melt viscosity Higher heat content (with heat of crystallization)

D.3.3 Amorphous Polymers

Molten polymer molecules in an unstressed state are randomly oriented and entangled with other molecules. Amorphous materials retain this type of entangled and disordered molecular configuration regardless of their states, as shown in Figure A.10.

D.3.3.1 Response to Temperature

When the temperature of melt decreases, amorphous polymers start becoming rubbery. When the temperature is further reduced to below the glass transition temperature, the amorphous polymers turn into glassy materials. Amorphous polymers possess a wide softening range (with no distinct melting temperature), moderate heat resistance, good impact resistance, and low shrinkage.

D.3.3.2 Differential Shrinkage in Thickness Direction

The molecules tend to be uncoiled and stretched in the flow direction as the cavity is filled. Those molecules that are quenched by contact with the cold mold wall will be frozen, stretched out in the flow direction. Molecules toward the interior of the part are insulated from the mold wall by the frozen layer. These will have time enough to recoil as they cool more slowly. That is, the molecules on the surface will be oriented and will shrink less; molecules in the interior will be less oriented and will shrink more. The differential shrinkage in the thickness direction results in flow-induced residual stresses in molded plastics.

D.3.3.3 Similar Linear Shrinkages

Families of amorphous plastics can often be substituted one for another, in the same injection cavities, since their linear shrinkages are in the same range. Therefore, styrene can be substituted for ABS; acrylics can be molded in the same cavities as polycarbonates. The properties will be different for the substitution, but the dimensions will usually be close enough to be within specified tolerances.

D.3.4 Semicrystalline Polymers

Crystalline materials are polymer chains that do not have bulky pendant groups, chain branches, or cross-links. They may accommodate themselves in a well-ordered regular lattice (polymer crystallite) when the molten polymers are cooled below the melting temperature, as shown in Figure A.10.

D.3.4.1 Controlling the Degree of Crystallinity

The crystallization process stops when the materials are cooled below the glass transition temperature. Since it is difficult to achieve 100% crystallization under normal processing conditions, any crystallizable polymers are typically semicrystalline, possessing both amorphous and crystalline phases. The degree of crystallinity depends on both the chemical structure of the polymer and the processing conditions. Crystalline and semicrystalline polymers have a distinct melting point, good chemical and heat resistance, good lubricity, low moisture absorption, and high shrinkage.

D.3.4.2 High Linear Shrinkage

The significantly higher linear shrinkage of the semicrystalline polymers precludes them being molded in the same cavities that are used for amorphous plastics: most dimensions will be significantly different and will most likely miss tolerances enough not to be functional in the same application.

D.3.4.3 Liquid Crystal Polymers

Liquid crystal polymers (LCPs) exhibit ordered molecular arrangements in both the melt and solid states, as shown in Figure A.10. These materials are characterized by their stiff, rodlike molecules that form the parallel arrays or domains. LCPs offer a number of processing and performance advantages including low melt viscosity, low mold shrinkage, chemical resistance, stiffness, creep resistance, and overall dimensional stability.

D.4 Thermosets

Cross-linking is a chemical process in which chemical bonds form among molecules of thermosetting materials, resulting in an interconnected network, as shown in Figure A.10. This cross-linking process is the principal difference between thermoplastics and thermosets. Thermosets inherently possess greater mechanical strength, higher service temperature limits, and greater dimensional stability than thermoplastics. Many thermosets are engineering resins and, because of the cross-linking, thermosets possess an amorphous structure.

D.4.0.1 Cross-linking (Reaction)

Prior to molding, the chainlike structure of thermosets is similar to thermoplastics. During processing, thermosets polymerize (react or cure) with the activation of heat and/or a chemical means into a cross-linked microstructure. Once the reaction is completed, the polymer chains are bonded (cross-linked) together to form a three-dimensional network. These cross bonds among molecules prohibit the slippage of individual molecular chains. Consequently, a thermoset becomes an infusible and insoluble solid and cannot be re-softened and reprocessed through the application of heat, without degrading some linkages. The best analogy to thermosets is that of a hard-boiled egg; the yolk has turned from a liquid to a solid and cannot be converted back to a liquid.

D.4.0.2 Processing Thermosets

Thermosets are usually purchased as liquid monomer-polymer mixtures or as a partially polymerized molding compound. Starting from this uncured condition, they can be formed to the final shape in the cavity by polymerization (activated either by heat or by chemical mixing) with or without pressure. Thermosets are generally filled or reinforced with materials (such as minerals, talc, or glass fibers) to impart specific properties (such as shrinkage control, chemical and shock resistance, electrical and thermal insulation) and/or to reduce cost.

D.5 Properties and Applications of Thermoplastics

This section reviews the general properties and typical applications of common thermoplastic resins used for injection molding. Generally, only injection-molded engineering applications are listed; packaging and textiles, for example, are not included.

Table 2: Typical melt and mold temperatures for various generic classes of resins

Generic Name	Melt Temperature (C/F)			Mold Temperature (C/F)			Ejection Temp (C/F)
	Min.	Rec.	Max.	Min.	Rec.	Max.	Rec.
ABS	200/392	230/446	280/536	25/77	50/122	80/176	88/190
PA 12	230/446	255/491	300/572	30/86	80/176	110/230	135/275
PA 6	230/446	255/491	300/572	70/158	85/185	110/230	133/271
PA 66	260/500	280/536	320/608	70/158	80/176	110/230	158/316
PBT	220/428	250/482	280/536	15/60	60/140	80/176	125/257
PC	260/500	305/581	340/644	70/158	95/203	120/248	127/261
PC/ABS	230/446	265/509	300/572	50/122	75/167	100/212	117/243
PC/PBT	250/482	265/509	280/536	40/104	60/140	85/185	125/257
HDPE	180/356	220/428	280/536	20/68	40/104	95/203	100/212
LDPE	180/356	220/428	280/536	20/68	40/104	70/158	80/176
PEI	340/644	400/752	440/824	70/158	140/284	175/347	191/376
PET	265/509	270/518	290/554	80/176	100/212	120/248	150/302
PETG	220/428	255/491	290/554	10/50	15/60	30/86	59/137
PMMA	240/464	250/482	280/536	35/90	60/140	80/176	85/185
POM	180/356	210/410	235/455	50/122	70/158	105/221	118/244
PP	200/392	230/446	280/536	20/68	50/122	80/176	93/199
PPE/PPO	240/464	280/536	320/608	60/140	80/176	110/230	128/262
PS	180/356	230/446	280/536	20/68	50/122	70/158	80/176
PVC	160/320	190/374	220/428	20/68	40/104	70/158	75/167
SAN	200/392	230/446	270/518	40/104	60/140	80/176	5/185

D.5.1 ABS

D.5.1.1 Generic Class

Acrylonitrile-Butadiene-Styrene

D.5.1.2 Typical Applications

- Automotive (instrument and interior trim panels, glove compartment doors, wheel covers, mirror housings, etc.)
- Refrigerators, small appliance housings and power tools applications (hair dryers, blenders, food processors, lawn mowers, etc.)
- Telephone housings, typewriter housings, typewriter keys
- Recreational vehicles (golf carts, jet skis, etc.)

D.5.1.3 Injection-molding Processing Conditions

Drying: ABS resins are hygroscopic and drying is required prior to processing. Suggested drying conditions are 80–90°C (176–195°F) for a minimum of 2 hours. Resin moisture content should be less than 0.1%.

Melt Temperature: 200–280°C (392–536°F); Aim: 230°C (446°F).

Mold Temperature: 25–80°C (77–176°F). (Mold temperatures control the gloss properties; lower mold temperatures produce lower gloss levels).

Injection Pressure: 50–100 MPa (7,250–14,500 psi).

Injection Speed: Moderate to high.

D.5.1.4 Chemical and Physical Properties

ABS is produced by a combination of three monomers: acrylonitrile, butadiene, and styrene. Each of the monomers impart different properties: hardness, chemical and heat resistance from acrylonitrile; processability, gloss, and strength from styrene; and toughness and impact resistance from butadiene. Morphologically, ABS is an amorphous resin.

The polymerization of the three monomers produces a terpolymer that has two phases: a continuous phase of styrene-acrylonitrile (SAN) and a dispersed phase of polybutadiene rubber. The properties of ABS are affected by the ratios of the monomers and molecular structure of the two phases. This allows a good deal of flexibility in product design and, consequently, there are hundreds of grades available in the market. Commercially available grades offer different characteristics, such as medium to high impact, low to high surface gloss, and high heat distortion.

ABS offers superior processability, appearance, low creep and excellent dimensional stability, and high impact strength.

D.5.1.5 Major Manufacturers

- BASF (Terluran)
- LANXESS (Lustran)
- Cheil Synthesis (Starex)
- Chi Mei (Polylac)
- Dow Chemical (Magnum)
- GE Plastics (Cycolac)
- LG Chemical (Lupos)

D.5.2 PA 12

D.5.2.1 Generic Class

Polyamide 12 or Nylon 12.

D.5.2.2 Typical Applications

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

D.5.2.3 Injection-molding Processing Conditions

Drying: The moisture content must be below 0.1% prior to processing.

If the material is exposed to air, drying in a hot air oven at 85°C (185°F) for 4 to 5 hours is recommended (3 to 4 hours in a desiccant dryer). If the container is unopened, it may be used directly for molding after 3 hours of equilibration to shop floor temperature.

Melt Temperature: 230–300°C (446–580°F); not to exceed 310°C (590°F) for standard grades and 270°C (518°F) for flame retardant grades.

Mold Temperature: 30–40°C (86–104°F) for unreinforced grades; for thin walled or large surface area components, 80–90°C (176–194°F) may be used; 90–100°C (194–212°F) for

reinforced grades. Increasing the mold temperature increases the crystallinity level. It is very important to precisely control the mold temperature.

Injection Pressure: Up to 100 MPa (14, 500 psi). Low hold pressures and high melt temperatures are recommended.

Injection Speed: High (high speeds give a better finish on glass-filled grades).

D.5.2.4 Runners and Gates

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

A variety of gates may be used. Small gates for large parts should not be used to avoid highly stressed components or excessive shrinkage. The thickness of the gate preferably should be equal to the part thickness. When using submarine gates, the minimum recommended diameter is 0.8 mm.

Hot runner molds may be used effectively, but precise temperature control is necessary to prevent material drooling or freezing off at the nozzle. When hot runners are used, the size of the gates may be smaller than in the case of cold runners.

D.5.2.5 Chemical and Physical Properties

PA 12 is a linear, semicrystalline-crystalline thermoplastic derived from butadiene. It has properties similar to PA 11, but its crystal structure is different. PA 12 is a good electrical insulator and its properties are not as sensitive to humidity as other polyamides. It has good resistance to shock and many chemicals. It is extensively modified with plasticisers and reinforcements. In comparison to PA 6 and PA 66, these materials have a lower melting point and density, and a much lower moisture regain. It is not resistant to strong oxidizing acids.

Viscosity is determined by water content, temperature, and residence time. This material flows easily. Shrinkage is of the order of 0.005 to 0.02 mm/mm (0.5 to 2%). This depends on the specific grade, wall thickness, and processing conditions.

D.5.2.6 Major Manufacturers

- Arkema (Rilsan)
- Degussa AG (Vestamid)
- PolyOne (Edgetek)

D.5.3 PA 6

D.5.3.1 Generic Class

Polyamide 6, Nylon 6, or Polycaprolactam

D.5.3.2 Typical Applications

Used in many structural applications because of its good mechanical strength and rigidity. It is used in bearings because of its good wear resistance.

D.5.3.3 Injection Molding Processing Conditions

Drying: Since PA 6 absorbs moisture readily, care should be taken to ensure its dryness prior to molding. If the material is supplied in watertight packaging, the containers should be kept closed. If the moisture content is $>0.2\%$, drying in a hot air oven at 80°C (176°F) for 16 hours is recommended. If the material has been exposed to air for more than 8 hours, vacuum drying at 105°C (221°F) for more than 8 hours is recommended.

Melt Temperature: $230\text{--}280^{\circ}\text{C}$ ($446\text{--}536^{\circ}\text{F}$); $250\text{--}300^{\circ}\text{C}$ ($482\text{--}572^{\circ}\text{F}$) for reinforced grades.

Mold Temperature: $80\text{--}90^{\circ}\text{C}$ ($176\text{--}194^{\circ}\text{F}$). 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 $80\text{--}90^{\circ}\text{C}$ ($176\text{--}194^{\circ}\text{F}$) are recommended. High mold temperatures are also recommended for thin-walled parts with long flow lengths. Increasing the mold temperature increases the strength and hardness, but the toughness is decreased. When the wall thickness is greater than 3 mm, a cold mold is recommended ($20\text{--}40^{\circ}\text{C}/68\text{--}104^{\circ}\text{F}$), which leads to a higher and more uniform degree of crystallinity. Glass reinforced resins are always processed at mold temperatures greater than 80°C (176°F).

Injection Pressure: Generally between 75 and 125 MPa (11,000 and 18,000 psi), depending on material and product design.

Injection Speed: High (slightly lower for reinforced grades).

D.5.3.4 Runners and gates

The gate location is important because of very fast freeze-off times. Any type of gate may be used; the aperture should not be less than half the thickness of the part. When hot runners are used, the size of the gates can be smaller than when cold runners are used because premature freeze-off is prevented. When using submarine gates, the minimum diameter of the gate should be 0.75 mm.

D.5.3.5 Chemical and Physical Properties

The molecular structure of polyamides consist of amide (CONH) groups joined by linear aliphatic sections (based on methylene groups). The toughness, rigidity, crystallinity, and thermal resistance of polyamide resins are because of the strong interchain attraction caused by the polarity of the amide groups. The CONH groups also cause a lot of moisture absorption.

Nylon 6 is produced by polymerization of caprolactam. The chemical and physical properties are similar to that of PA 66. However, its melting point is lower than PA 66 and it has a wider processing temperature range. Its impact strength and solvent resistance are better than PA 66, but its moisture absorption is higher. Many properties are affected by moisture absorption, which must be taken into account when designing with this resin. Various modifiers are added to improve mechanical properties; glass is one of the most commonly used fillers. The addition of elastomers such as EPDM or SBR improves impact resistance.

For unfilled grades, shrinkage is of the order of 0.01 to 0.015 mm/mm (1 to 1.5%). The addition of glass fibers reduce the shrinkage to as low as 0.3% in the flow direction (but could be as high as 1% in the cross-flow direction). The post-molding shrinkage is affected mainly by the crystallinity level and moisture absorption. The actual shrinkage is a function of part design, wall thickness, and processing conditions.

D.5.3.6 Major Manufacturers

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

D.5.4 PA 66

D.5.4.1 Generic Class

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

D.5.4.2 Typical Applications

PA66 competes with PA 6 for most applications. PA 66 is heavily used in the following:

- The automotive industry
- Appliance housings
- Where impact resistance and strength are required

D.5.4.3 Injection-molding Processing Conditions

Drying: Drying is not required if the material is sealed prior to molding; however, if the containers are left open, drying in a hot air oven at 85°C (185°F) is recommended. If the moisture content is > 0.2%, vacuum drying at 105°C (220°F) for 12 hours is recommended.

Melt Temperature: 260–290°C (500–554°F); 275–280°C (527–536°F) for glass filled grades. Melt temperatures above 300°C (572°F) should be avoided.

Mold Temperature: 80°C (176°F) suggested. Mold temperature affects crystallinity level, which in turn affects physical properties. In the case of thin-walled parts, crystallinity changes with time if mold temperatures of less than 40°C (104°F) are used. In such cases, annealing may be needed to retain dimensional stability.

Injection Pressure: Generally between 75 and 125 MPa (11,000 and 18,000 psi), depending on material and product design.

Injection Speed: High (slightly lower for reinforced grades).

D.5.4.4 Runners and Gates

The gate location is important because of very fast freeze-off times. Any type of gate may be used; the aperture should not be less than half the thickness of the part. When hot runners are used, the size of the gates can be smaller than when cold runners are used, because premature freeze-off is prevented. When using submarine gates, the minimum diameter of the gate should be 0.75 mm.

D.5.4.5 Chemical and Physical Properties

PA 66 homopolymer is produced by the polymerization of hexamethylene diamine and adipic acid (a dibasic acid). Among commercially available polyamides, PA 66 has one of the highest melting points. It is a semicrystalline-crystalline material. The resins have strength and stiffness that is retained at elevated temperatures. It does absorb moisture after molding, but the retention is not as much as in the case of PA 6. Moisture absorption depends on the composition of the material, wall thickness, and environmental conditions. Dimensional stability and properties are all affected by the amount of moisture absorption which must be taken into account for product design.

Various modifiers are added to improve mechanical properties; glass is one of the most commonly used fillers. The addition of elastomers such as EPDM or SBR improves impact resistance.

The viscosity is low and, therefore, it flows easily (but not as easily as PA 6). This allows molding of thin components. The viscosity is very sensitive to temperature. Shrinkage is of the order of 0.01 to 0.02 mm/mm (1 to 2%). The addition of reinforcing glass fibers reduces the shrinkage to 0.2 to 1%. Differential shrinkage in the flow and cross-flow directions is quite high. Mineral fillers yield more isotropic moldings. PA 66 is resistant to most solvents but not to strong acids or oxidizing agents.

D.5.4.6 Major Manufacturers

- BASF (Ultramid A)
- DSM (Akulon)
- DuPont (Zytel)
- Solutia (Vydyne)

D.5.5 PBT

D.5.5.1 Generic Class

Polybutylene Terephthalates

D.5.5.2 Typical Applications

- Household appliances (e.g., food processor blades, vacuum cleaner parts, fans, hair dryer housings, coffee makers)
- Electronics (e.g., switches, motor housings, fuse cases, key caps for computer keyboards, connectors, fiber-optic buffer tubing)
- Automotive (e.g., grilles, body panels, wheel covers, and components for doors and windows)

D.5.5.3 Injection-molding Processing Conditions

Drying: This material is sensitive to hydrolysis at high temperatures. It is therefore important to dry the material prior to molding. Suggested drying conditions (in air) are 120°C (248°F) for 6 to 8 hours [or 150°C (300°F) for 2 to 4 hours]. Moisture levels must be below 0.03%. When using a desiccant dryer, drying at 120°C (248°F) for 2.5 hours is recommended.

Melt Temperature: 220–280°C (428–536°F); aim: 250°C (482°F).

Mold Temperature: 40–60°C (104–140°F) for unreinforced grades. For other grades, a wide range of temperatures can be used, depending on the grade (15–80°C/59–176°F). Cooling channels should be properly designed to minimize part warpage. The heat removal must be fast and uniform. Cooling channels of 12 mm diameter are recommended.

Injection Pressure: Moderate (up to maximum of 150 MPa/21750 psi).

Injection Speed: Fastest possible speeds should be used (due to fast solidification of PBTs).

D.5.5.4 Runners and Gates

Full round runners are recommended to impart maximum pressure transmission (rule of thumb: runner diameter = part thickness + 1.5 mm). A wide variety of gates may be used. Hot runners may also be used, taking care to avoid drool and material degradation. Gate diameters or depths should preferably be between 0.8 to 1.0 times the part thickness. When using submarine gates, the minimum recommended diameter is 0.75 mm.

D.5.5.5 Chemical and Physical Properties

PBT is one of the toughest engineering thermoplastics. It is a semicrystalline resin and has excellent chemical resistance, mechanical strength, electrical properties (high dielectric strength and insulation resistance), and heat resistance, all of which are stable over a broad range of environmental conditions. It has very low moisture absorption.

PBT, which is a polyester, is produced by the polycondensation reaction of dimethyl terephthalate 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, which could cause warpage from non-uniform cooling. In the case of glass-filled grades, shrinkage is reduced in the flow direction, but in the cross-flow direction it may be equal to that of the base resin. Shrinkage is on the order of 0.015 to 0.028 mm/mm (1.5 to 2.8%). A 30% glass-filled resin has a shrinkage range of 0.3 to 1.6%. The melting point (approximately 225°C/437°F) and heat distortion temperatures are lower than that of PET. The Vicat softening point is approximately 170°C (338°F). The glass transition temperature ranges from 22 to 43°C (71 to 109°F).

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

D.5.5.6 Major Manufacturers

- BASF (Ultradur)
- LANXESS (Pocan)
- GE Plastics (Valox)
- Ticona (Celanex)

D.5.6 PC

D.5.6.1 Generic Class

Polycarbonate

D.5.6.2 Typical Applications

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

D.5.6.3 Injection-molding Processing Conditions

Drying: PC resins are hygroscopic and pre-drying is important. Recommended drying conditions are 100 to 120°C (212 to 248°F) for 3 to 4 hours. Moisture content must be less than 0.02% prior to processing.

Melt Temperature: 260–340°C (500–644°F); higher range for low MFR resins and vice-versa.

Mold Temperature: 70–120°C (158–248°F); higher range for low MFR resins and vice-versa.

Injection Pressure: As high as possible for rapid molding.

Injection Speed: Slow injection speeds when small or edge gates are used; high speeds for other types of gates.

D.5.6.4 Chemical and Physical Properties

Polycarbonate is a polyester of carbonic acid. All general-purpose polycarbonates are based on bisphenol A. The bisphenol A component of the molecule contributes to the high glass transition temperature (150°C/302°F). The rotational mobility of the carbonyl group within the molecule contributes to the high ductility and toughness of the resin.

PC is an amorphous engineering resin with exceptionally good impact strength, heat resistance, clarity, sterilizability, flame retardancy, and stain resistance. The notched Izod impact strength of PC is very high and mold shrinkage is low and consistent (0.1 to 0.2 mm/mm).

High-molecular-weight PCs (which translates to low melt flow rate) have higher mechanical properties, but processability of such resins becomes difficult. The type of PC chosen for a particular application should be based on the desired criteria (for high impact properties, use a low-MFR PC; conversely, for optimal processability, use a high-MFR PC).

The melt viscosities are typically Newtonian up to shear rates of 1000 1/s and decrease beyond that. The Heat Deflection Temperature Under Load is typically between 130 and 140°C (266 and 284°F) and the Vicat Softening Point is typically around 155°C (311°F).

D.5.6.5 Major Manufacturers

- Bayer (Apec, Makrolon)
- Dow Chemical (Calibre)
- DSM (Xantar)
- GE Plastics (Lexan)
- LNP (Thermocomp)
- Teijin Chemical (Panlite)

D.5.7 PC/ABS

D.5.7.1 Generic Class

Polycarbonate | Acrylonitrile-Butadiene-Styrene Blend

D.5.7.2 Typical Applications

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

D.5.7.3 Injection-molding Processing Conditions

Drying: Drying is required prior to processing. Moisture content should be less than 0.04% to ensure stable processing parameters. Drying at 90 to 110°C (194 to 230°F) for 2 to 4 hours is recommended.

Melt Temperature: 230–300°C (446–572°F).

Mold Temperature: 50–100°C (122–212°F).

Mold Temperature: Part dependent.

Injection Pressure: Part dependent.

Injection Speed: As high as possible.

D.5.7.4 Chemical and Physical Properties

PC/ABS offers combined properties of PC and ABS (high processability of ABS along with excellent mechanical properties and impact and heat resistance of PC). The ratio of the two components affects the heat resistance. The blend exhibits excellent flow characteristics.

D.5.7.5 Major Manufacturers

- Bayer (Bayblend)
- Dow Chemicals (Pulse)
- DSM (Xantar)
- GE Plastics (Cycloy)
- Teijin Chemical (Multilon)

D.5.8 PC/PBT

D.5.8.1 Generic Class

Polycarbonate | Polybutyleneterephthalate Blend

D.5.8.2 Typical Applications

- Gear cases and automotive (bumpers)
- Applications that require chemical and corrosion resistance, high heat resistance, high impact strength over wide temperature ranges, and high dimensional stability

D.5.8.3 Injection-molding Processing Conditions

Drying: 110–135°C (230–275°F) for approximately 4 hours.

Melt Temperature: 230–280°C (402–536°F) depending on specific grade.

Mold Temperature: 40–85°C (104–185°F).

D.5.8.4 Chemical and Physical Properties

PC|PBT blends offer a combination of properties of PC and PBT: high toughness and dimensional stability of PC and good chemical and heat resistance and lubricity of crystalline PBT.

D.5.8.5 Major Manufacturers

- MRC Polymers (Naxaloy)
- GE Plastics (Xenoy)

D.5.9 HDPE

D.5.9.1 Generic Class

High Density Polyethylene

D.5.9.2 Typical Applications

Major use is in blow-molding (packaging) applications such as:

- Containers in refrigeration units
- Storage vessels
- Household goods (kitchenware)
- Seal caps
- Bases for PET bottles

D.5.9.3 Injection-molding Processing Conditions

Drying: Not normally necessary if stored properly.

Melt Temperature: 180–280°C (356–536°F). For high-molecular-weight resins, the suggested melt temperature range is 200 to 250°C (392 to 482°F).

Mold Temperature: 20–95°C (68–194°F), with higher temperatures for wall thickness of up to 6 mm and lower temperatures for wall thicknesses greater than 6 mm.

Injection Pressure: 70–105 MPa (10,000–15,000 psi)

Injection Speed: Fast injection speeds are recommended; profiled speeds reduce warpage in the case of components with a large surface area.

D.5.9.4 Runners and Gates

Diameters of runners range from 4 to 7.5 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.

D.5.9.5 Chemical and Physical Properties

High-density polyethylene is produced from polymerization of ethylene (lower temperature and pressure conditions are used compared to the production of low-density polyethylene). The material is free from branching, which is made possible by the use of stereospecific catalysts. Because of molecular regularity, HDPE has a high level of crystallinity (compared to LDPE).

Higher levels of crystallinity contribute to higher density, tensile strength, heat distortion temperature, viscosity, and chemical resistance. HDPE is more resistant to permeability than LDPE. The impact strength is lower. The properties of HDPE are controlled by the density, and molecular weight distributions. Injection molding grades typically have a narrow molecular weight distribution.

When the density is 0.91 to 0.925 g/cm³, the material is known as Type 1; Type 2 materials have densities in the range of 0.926 to 0.94 g/cm³; and Type 3 materials have densities in the range of 0.94 to 0.965 g/cm³.

The material flows easily and the MFR ranges from 0.1 to 28. Higher molecular weights (lower MFR grades) have better impact resistance. Being a semicrystalline material, the molding shrinkage is high (order of 0.015 to 0.04 mm/mm or 1.5 to 4%). This depends on the degree of orientation and level of crystallinity in the part (which in turn depend on processing conditions and part design).

PE is susceptible to environmental stress cracking, which can be minimized by reducing internal stresses by proper design and using the lowest MFR material at a particular density level. HDPE is soluble in hydrocarbons at temperatures greater than 60°C, but resistance to these materials is greater than that for LDPE.

D.5.9.6 Major Manufacturers

- ExxonMobil (Escorene, Paxon)
- Basell (Lupolen)
- Dow (Dowlex)
- Arkema (Lacqtene)
- Equistar (Alathon)
- Chevron Phillips (Marlex)
- Nova Chemicals (Sclair)

D.5.10 LDPE

D.5.10.1 Generic Class

Low Density Polyethylene

D.5.10.2 Typical Applications

- Closures
- Bowls
- Bins
- Pipe couplings

D.5.10.3 Injection Molding Processing Conditions

Drying: Not usually necessary.

Melt Temperature: 180–280°C (355–535°F).

Mold Temperature: 20–70°C (68–158°F); for uniform and economic heat removal, it is recommended that the cooling channel diameters be at least 8 mm and the distance from the surface of the mold to the edge of the cooling channel be not more than 1.5 times the diameter of the cooling channel.

Injection Pressure: Up to 150 MPa (21,750 psi).

Injection Speed: Fast speeds are recommended; profiled speeds can limit warpage problems of large surface area parts.

D.5.10.4 Runners and Gates

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

D.5.10.5 Chemical and Physical Properties

Low density polyethylene is produced by the polymerization of ethylene at high pressure and temperature. The resin is semicrystalline-crystalline. The crystallinity level is low because of chain branching. The material is tough but possesses moderate tensile properties and exhibits creep. However, it has good impact and chemical resistance. It is an easy flow material because of long chain branching.

Commercial materials have densities in the range of 0.91 to 0.94 g/cm³. LDPE is permeable to gases and vapors. Very close tolerances are not possible with this resin and its relatively large coefficient of thermal expansion makes it less suitable for long term applications.

Shrinkage is of the order of 0.02 to 0.05 mm/mm (2 to 5%) when density is between 0.91 to 0.925 g/cm³. When density is between 0.926 to 0.940 g/cm³, the shrinkage is of the order of 1.5 to 4%. Actual shrinkage values depend on the molding conditions.

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

D.5.10.6 Major Manufacturers

- Arkema (Lacqtene)
- Basell (Lupolen)
- Chevron Phillips
- Dow (Dowlex LDPE)
- Nova Chemicals (Novapol)
- Quantum Chemicals

D.5.11 PEI

D.5.11.1 Generic Class

Polyetherimide

D.5.11.2 Typical Applications

- Automotive (engine components: temperature sensors, fuel and air handling devices)
- Electrical/electronics (connector materials, printed circuit boards, circuit chip carriers, explosion proof boxes)
- Packaging applications
- Aircraft (interior materials)
- Medical (surgical staplers, tool housings, nonimplant devices)

D.5.11.3 Injection-molding Processing Conditions

Drying: PEI absorbs moisture and can cause material degradation. Moisture content should be less than 0.02%. Suggested drying conditions are 150°C (302°F) for 4 hours in a desiccant dryer (6 hours for reinforced and blended grades).

Melt Temperature: 340–400°C (644–752°F) for unreinforced grades; 340–440°C (644–824°F) reinforced grades.

Mold Temperature: 70–175°C (108–347°F). Aim: 140°C.

Injection Pressure: 7–150 MPa (10,000–22,000 psi) typical.

Injection Speed: As high as possible.

D.5.11.4 Chemical and Physical Properties

PEIs are amorphous materials whose chemical structure consists of repeating aromatic imide and ether units. This accounts for its high temperature resistance. It also leads to high stiffness, and modifiers are used to make the material processable. PEIs are very stiff and strong even without reinforcements. They have excellent thermal stability making it possible to use them in high temperature applications. They have good flame and chemical resistance and good electrical insulation properties. The glass transition temperature is high (215°C/419°F). It exhibits low shrinkage and highly isotropic mechanical properties.

D.5.11.5 Major Manufacturers

- GE Plastics (Ultem)
- RTP Company

D.5.12 PET

D.5.12.1 Generic Class

Polyethylene Terephthalate

D.5.12.2 Typical applications

- Automotive (structural components, such as mirror backs and grille supports; electrical parts, such as head lamp reflectors and alternator housings)
- Electrical applications (motor housings, electrical connectors, relays, and switches, microwave oven interiors)
- Industrial applications (furniture chair arms, pump housings, hand tools)

D.5.12.3 Injection Molding Processing Conditions

Drying: Drying is essential prior to molding. PETs are very sensitive to hydrolysis. Recommended drying conditions are 120 to 165°C (248 to 329°F) for 4 hours. The moisture content should be less than 0.02%.

Melt Temperature: 265–280°C (509–536°F) for unfilled grades; 275–290°C (527–554°F) for glass-reinforced grades.

Mold Temperature: 80–120°C (176–248°F). Preferred range: 100–110°C; (212–230°F).

Injection Pressure: 300–130 MPa (4,350–19,000 psi).

Injection Speed: High speeds without causing embrittlement.

D.5.12.4 Runners and Gates

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

D.5.12.5 Chemical and Physical Properties

PET is an aromatic polyester produced from polymerization of either terephthalic acid (TPA) or dimethyl ester terephthalic acid (DMT) and ethylene glycol (EG). The glass transition is approximately 165°C (330°F) and the resin crystallizes over a temperature range from 120 to 220°C (248 to 428°F).

PET is highly sensitive to moisture at high temperatures and exhibits excessive warpage when reinforced with glass fibers. Promotion of crystallinity is achieved through adding nucleating agents and crystal growth accelerators. Crystalline moldings exhibit high modulus, gloss, and heat distortion temperatures. Warpage is minimized by addition of particulate fillers such as mica. When low mold temperatures are used, clear moldings can be obtained with unfilled PETs.

D.5.12.6 Major Manufacturers

- BASF (Petra)
- DuPont (Rynite)
- Eastman Chemical (Eastapak)
- Ticona (Impet)

D.5.13 PETG

D.5.13.1 Generic Class

Glycol-modified PET; Copolyesters

D.5.13.2 Typical Applications

PETGs offer a desirable combination of properties such as clarity, toughness, and stiffness. Applications include:

- Medical devices (test tubes and bottles)
- Toys
- Displays
- Lighting fixtures
- Face shields
- Refrigerator crisper pans

D.5.13.3 Injection-molding Processing Conditions

Drying: Drying is essential for PETG prior to injection molding. The moisture level must be below 0.04%. Drying temperature is not to exceed 66°C (150°F). Drying at approximately 65°C (149°F) for 4 hours is recommended.

Melt Temperature: 220–290°C (428–554°F). The melt temperature is grade specific.

Mold Temperature: 10–30°C (50–86°F). Recommended: 15°C (60°F).

Injection Pressure: 30–130 MPa (4,350–19,000 psi)

Injection Speed: High speeds without causing embrittlement.

D.5.13.4 Chemical and Physical Properties

PETGs (or copolyesters) are glycol modified PETs; the modification is done by adding a second glycol during polymerization. The resulting molecular structure is irregular and the resin is clear and amorphous with a glass transition temperature of 88°C (190°F). PETGs can be processed over a wider processing range than conventional PETs and offer good combination of properties such as toughness, clarity, and stiffness.

D.5.13.5 Major Manufacturers

- Eastman (Eastar)
- Noveon (Stat-Rite)
- RTP Company (PermaStat)

D.5.14 PMMA

D.5.14.1 Generic Class

Polymethyl Methacrylate

D.5.14.2 Typical Applications

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

D.5.14.3 Injection-molding Processing Conditions

Drying: PMMA is hygroscopic and must be dried prior to molding. Drying at 90°C (194°F) for 2 to 4 hours is recommended.

Melt Temperature: 240–280°C (460–536°F).

Mold Temperature: 35–80°C (90–176°F).

Injection Speed: Moderate.

D.5.14.4 Chemical and Physical Properties

Pellets for injection molding are made either by bulk polymerization of methyl methacrylate followed by extrusion and pelletization or by polymerization in an extruder. Formulations vary by molecular weight and physical properties such as flow rate, heat resistance, and toughness. Higher molecular weight grades are tougher than lower molecular weight grades. High flow formulations are generally preferred for molding.

Heat deflection temperature under load varies from 75°C (167°F) for high flow materials to 100°C (212°F) for low flow (high molecular weight) materials.

PMMA has excellent optical properties and weatherability. The white light transmittance is as high as 92%. Molded parts can have very low birefringence, which makes it ideally suited as a material for video discs.

PMMA exhibits room temperature creep. The initial tensile strength is high but under long term, high stress loading, it exhibits stress craze. Impact strength is good but it does show some notch sensitivity.

D.5.14.5 Major Manufacturers

- Arkema/Atoglas (Plexiglas)
- BASF (Lucryl)
- Degussa (Plexiglas)
- Cyro Industries (Acrylite, Cyrolite)
- LG Chemical

D.5.15 POM

D.5.15.1 Generic Class

Polyacetal or Polyoxymethylene

D.5.15.2 Typical Applications

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

D.5.15.3 Injection Molding Processing Conditions

Drying: Not usually required but resin should be stored in a dry atmosphere.

Melt Temperature: 190–230°C (374–446°F) for homopolymer; 180–210°C (356–410°F) for copolymer.

Mold Temperature: 80–105°C (122–221°F). Higher mold temperatures are preferred for precision molding to reduce post-molding shrinkage.

Injection Pressure: 70–120 MPa (10,000–17,500 psi).

Injection Speed: Medium to high.

D.5.15.4 Runners and Gates

Any type of gate may be used. When using tunnel gates, the short type is preferred. Insulated, hot tip runners are preferred for homopolymers; both internally and externally heated hot runners may be used in the case of copolymers.

D.5.15.5 Chemical and Physical Properties

Acetals are tough, resilient materials and exhibit good creep resistance, dimensional stability, and impact resistance even at low temperatures. Acetal resins are either homopolymers or copolymers. Homopolymers have better tensile strength, fatigue resistance, and hardness but are difficult to process. Copolymers have better thermal stability, chemical resistance, and processability. Both homopolymers and copolymers are crystalline and have low moisture absorption.

Copolymers may be used continuously at air temperatures up to 100°C (212°F); homopolymers have slightly higher temperature resistance. Many grades of acetal resins are available, tailored to different applications.

High crystallinity levels of acetals lead to relatively high shrinkage levels of 0.02 to 0.035 mm/mm. Differential shrinkage is observed with reinforced grades.

D.5.15.6 Major Manufacturers

- BASF (Ultraform copolymers)
- DuPont (Delrin homopolymers)
- Ticona (Hostaform, Celcon copolymers)

D.5.16 PP

D.5.16.1 Generic Class

Polypropylene

D.5.16.2 Typical Applications

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

D.5.16.3 Injection-molding Processing Conditions

Drying: Not normally necessary if proper storage is used.

Melt Temperature: 220–280°C (428–536°F) not to exceed 280°C.

Mold Temperature: 20–80°C (68–176°F). Suggested: 50°C (122°F). The crystallinity level is determined by the mold temperature.

Injection Pressure: Up to 180 MPa (26,000 psi).

Injection Speed: Typically, fast injection speeds are used to minimize internal stresses; if surface defects occur, slow speed molding at a higher temperature is preferred. Machines capable of providing profiled speed are highly recommended.

D.5.16.4 Runners and Gates

In the case of cold runners, typical diameters range from 4 to 7 mm. Fully round sprues and runners are recommended. All types of gates can be used. Typical pin gate diameters range from 1 to 1.5 mm, but diameters as low as 0.7 mm may be used. In case of edge gating, the minimum gate depth should be half the wall thickness and the width should be at least double the thickness. Hot runners can readily be used for molding PP.

D.5.16.5 Chemical and Physical Properties

PP is produced by the polymerization of propylene using stereospecific catalysts. Mainly, isotactic PP is produced (the methyl groups lie on one side of the carbon chain). This linear plastic is semicrystalline because of ordered molecular structure. It is stiffer than PE and has a higher melting point. The PP homopolymer becomes very brittle at temperatures higher than 0°C (32°F) and for this reason, many commercially available grades are random copolymers with 1 to 4% ethylene or block copolymers with higher ethylene content. Copolymers have a lower heat distortion temperature (approximately 100°C/212°F), less clarity, gloss, and rigidity, but greater impact strength. The material becomes tougher as the ratio of ethylene increases. The Vicat softening point is approximately 150°C (302°F). Because of high levels of crystallinity, the surface hardness and scratch resistance is higher for these materials.

PP does not have environmental stress cracking problems. PP is usually modified by addition of glass fibers, mineral fillers, or thermoplastic rubbers. The MFR of PP ranges from 1 to 40; lower MFR materials have better impact strength but lower tensile strength. The copolymer is tougher than the homopolymer of the same MFR. The viscosity is more shear and temperature sensitive than PE.

Due to crystallinity, the shrinkage is relatively high (order of 0.018 to 0.025 mm/mm or 1.8 to 2.5%). The shrinkage is more uniform than PE-HD (the difference in flow and cross-flow shrinkage is typically less than 0.2%). Addition of 30% glass reduces the shrinkage to approximately 0.7%.

Both homopolymer and copolymer PP offer excellent resistance to moisture and good chemical resistance to acids, alkalis, and solvents. However, it is not resistant to aromatic hydrocarbons such as benzene, and chlorinated hydrocarbons such as carbon tetrachloride. It is not as resistant to oxidation at high temperatures as PE.

D.5.16.6 Major Manufacturers

- Albis
- Amoco
- Arkema (Appryl)
- A. Schulman (PolyFort)
- BASF (Novolen)
- Borealis
- Ebbtide Polymers (Aclo Accutech)
- Exxon Chemical
- Montel
- Phillips Sumika (Marlex)

D.5.17 PPE/PPO

D.5.17.1 Generic Class

Polypropylene Ether Blends

D.5.17.2 Typical Applications

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

D.5.17.3 Injection Molding Processing Conditions

Drying: Recommend drying before molding for approximately 2 to 4 hours at 100°C (212°F). PPOs have low levels of moisture absorption and can typically be molded as received.

Melt Temperature: 240–320°C (464–608°F); higher ranges for resins with higher levels of PPO.

Mold Temperature: 60–105°C (140–220°F).

Injection Pressure: 60–150 MPa (8,700–21,750 psi).

D.5.17.4 Runners and Gates

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

D.5.17.5 Chemical and Physical Properties

PPO is poly (2,6 dimethyl p-phenylene) oxide. The ether linkages offer easier processability. Copolymers are referred to as PPEs (Polyphenylene Ethers). Typically, the commercially available PPOs (or PPEs) are blended with other thermoplastic materials such as PS (or HIPS), Nylon, etc. These blends are still referred to as PPOs or PPEs.

The blends offer superior processability compared to pure PPOs. Their viscosities are lower. A range of properties can be obtained depending on the ratios of PPO and PS. Blends with nylons (PA 6/6) offer improved chemical resistance and perform well at high temperatures. The water absorption is low and the molded products have excellent dimensional stability.

Blends with PS are amorphous whereas blends with Nylons are crystalline. The addition of glass fibers reduces shrinkage levels to 0.2%. These materials have excellent dielectric properties and a low coefficient of thermal expansion. The viscosity level depends on the ratio of the components in the blend; higher PPO levels increase the viscosity.

D.5.17.6 Major Manufacturers

- Custom Resins (Norpex)
- Degussa (Vestoran)
- GE Plastics (Noryl)

D.5.18 PS

D.5.18.1 Generic Class

Polystyrene

D.5.18.2 Typical Applications

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

D.5.18.3 Injection-molding Processing Conditions

Drying: Not usually required unless stored improperly. If drying is needed, the recommended conditions are 2 to 3 hours at 80°C (176°F).

Melt Temperature: 180–280°C (356–536°F). The upper limit is 250°C (482°F) for flame retardant grades.

Mold Temperature: Suggested: 20–70°C (68–158°F).

Injection Pressure: 20–60 MPa (3,000–8,700 psi).

Injection Speed: Fast speeds are recommended.

D.5.18.4 Runners and Gates

All types of conventional gates may be used.

D.5.18.5 Chemical and Physical Properties

General-purpose PS is produced by the polymerization of styrene. Most commercial grades are clear, amorphous polymers. PS offers excellent dimensional and thermal stability, optical clarity, and very little tendency to absorb moisture. It has good dielectric properties. It is resistant to water and dilute inorganic acids, but is attacked by strong oxidizing acids such as concentrated sulfuric acid, and is swollen by some organic solvents.

Processing shrinkage is typically between 0.4 and 0.7%.

D.5.18.6 Major Manufacturers

- Albis
- Arkema (Lacqrene)
- BASF (Polystyrol)
- Chevron
- Dow (Questra)
- Creanova (Vestyron)

D.5.19 PVC

D.5.19.1 Generic Class

Polyvinyl Chloride

D.5.19.2 Typical Applications

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

D.5.19.3 Injection-molding Processing Conditions

Drying: Not usually necessary, as PVC absorbs very little water.

Melt Temperature: 160–220°C (320–428°F).

Mold Temperature: 20–70°C (68–158°F).

Injection Pressure: Up to 150 MPa (21,750 psi).

Packing Pressure: Up to 100 MPa (14,500 psi).

Injection Speed: Relatively slow, to avoid material degradation.

D.5.19.4 Runners and Gates

All conventional gate types may be used; pin-point and submarine gates are used for molding small components and fan gates are typically used for thick sections. The minimum diameter of pin-point or submarine gates should be 1 mm and the thickness of fan gates should not be less than 1 mm.

Sprues should be as short as possible; typical runner sizes are 6 to 10 mm and should have a full round cross-section. Insulated hot runners and certain types of hot sprue bushings may be used with PVC.

D.5.19.5 Chemical and Physical Properties

Rigid (unplasticized) PVC is one of the most widely used plastic materials. It is produced from sodium chloride and natural gas. The repeat chemical structure is vinyl chloride. Additives are mixed with PVC to make it processable. PVC resins produced by suspension or mass polymerization techniques are the major types used for melt processing. PVC is substantially an amorphous material.

Some of the additives used include stabilizers, lubricants, processing aids, pigments, impact modifiers, and fillers. Some features of PVC include low combustibility, toughness (designed to be virtually unbreakable), good weatherability (including good color and impact retention, and no loss in stiffness), and excellent dimensional stability. PVC is highly resistant to oxidizing and reducing agents, and strong acids. However, unplasticized PVC is not recommended for environmental and continuous use above 60°C (140°F). It is not resistant to concentrated oxidizing acids such as sulfuric or nitric acid and is unsuitable for use with aromatic and chlorinated hydrocarbons.

It is very important to process the material at the correct melt temperature; otherwise severe problems from decomposition (which produces hydrochloric acid which in turn accelerates decomposition) could result.

PVC is a relatively stiff flow material and has a narrow processing range. The molecular weight determines the flow characteristics. Higher molecular weight materials are more difficult to process (this could be modified by addition of lubricants). Typically, however, relatively low molecular weight resins are used in molding. Shrinkage is fairly low (0.002 to 0.006 mm/mm or 0.2 to 0.6%).

D.5.19.6 Major Manufacturers

- Arkema (Lacovyl)
- BASF (Vinidur, Vinoflex)
- Creanova (Vestolit)
- Keysor-Century
- PolyOne (Geon)
- Teknor APEX

D.5.20 SAN

D.5.20.1 Generic Class

Styrene Acrylonitrile

D.5.20.2 Typical Applications

- Electrical (receptacles, mixer bowls, housings, etc. for kitchen appliances; refrigerator fittings; chassis for television sets; cassette boxes)
- Automotive (head lamp bodies, reflectors, glove compartments, instrument panel covers)
- Household appliances (tableware, cutlery, beakers)
- Cosmetic packs

D.5.20.3 Injection Molding Processing Conditions

Drying: Under improper storage conditions, SAN absorbs moisture; it is recommended that it be dried at 80°C (176°F) for 2 to 4 hours prior to molding.

Melt Temperature: 200–270°C (392–518°F). For most applications, 230–260°C (446–500°F); the lower end of the range is used for molding thick-walled components.

Mold Temperature: 40–80°C (104–176°F). SAN solidifies rapidly at higher temperatures; in case of reinforced grades, the mold temperatures should not be less than 60°C (140°F). The cooling system must be well designed because the mold temperature affects the parts appearance and shrinkage and warpage.

Injection Pressure: 35–130 MPa (5,000–20,000 psi).

Injection Speed: High speeds are recommended.

D.5.20.4 Gates

All conventional gate types may be used. The gates must be of proper size to aid in processing without causing streaks, burn marks, or voids.

D.5.20.5 Chemical and Physical Properties

SAN copolymers are produced by the polymerization reaction of styrene and acrylonitrile. They are strong, transparent materials. The styrene component imparts clarity, stiffness, and processability and the acrylonitrile component imparts chemical and thermal resistance.

They have excellent load bearing capacity and rigidity; good resistance to chemicals, heat deformation, and cyclic temperature loads; and dimensional stability. The properties depend on the acrylonitrile content and commercial grades offer different acrylonitrile molecular masses. The addition of glass fibers enhances rigidity and resistance to heat deformation, and decreases the coefficient of linear thermal expansion.

The Vicat softening point is approximately 110°C (230°F) and the deflection temperature under load is approximately 100°C (212°F).

Shrinkage ranges from 0.003 to 0.007 mm/mm (0.3 to 0.7%).

D.5.20.6 Major Manufacturers

- BASF (Luran)
- Dow Chemical (Tyril)
- LANXESS (Lustran)
- Network Polymers

D.5.21 Additives, Fillers, and Reinforcements

Additives, fillers, and reinforcements are used to change and improve the physical and mechanical properties of plastics. In general, reinforcing fibers increase the mechanical properties of polymer composites while particulate fillers of various types increase the modulus.

Table 3 lists a variety of additives, fillers, and reinforcements and their effects on the polymer properties.

Table 3: Effects of additives, fillers, and reinforcements on polymer properties

Additive/Filler/Reinforcement	Common Materials	Effects on Polymer Properties
Reinforcing fibers	Boron Carbon Fibrous minerals Glass Kevlar	Increases tensile strength. Increases flexural modulus Increased heat-deflection temperature (HDT) Resists shrinkage and warpage
Conductive fillers	Aluminum powders Carbon fiber Graphite	Improves electrical and thermal conductivity
Coupling agents	Silanes Titanates	Improves interface bonding between polymer matrix and fibers
Flame retardants	Chlorine Bromine Phosphorous Metalic salts	Reduced the occurrence and spread of combustion
Extender fillers	Calcium carbonate Silica Clay	Reduces material cost
Plasticizers	Monomeric liquids Low-molecular weight materials	Improves melt flow properties Enhances flexibility
Colorants (pigments and dyes)	Metal oxides Chromates Carbon blacks	Provides color fastness Protects from thermal and UV degradation (with carbon blacks)
Blowing agents	Gas Azo compounds Hydrazine derivatives	Generates a cellular form to obtain a low-density material

D.5.22 Modifying Polymer Properties

Electrical properties can be affected by many fillers. For example, by adding conductive fillers an electromagnetic shielding property can be built into plastics, which are normally poor electrical conductors. Antistatic agents can be used to attract moisture, reducing the build-up of static charge.

Coupling agents are added to improve the bonding of the plastic matrix and the reinforcing fibers. Different fillers are used to lower the cost of materials. Other additives include flame retardants to reduce the likelihood of combustion, lubricants to reduce the viscosity of the molten plastic, plasticizers to increase the flexibility of the materials, and colorants to provide colorfastness.

D.5.23 Low-aspect Fillers

Fillers modify the properties and molding of the compound to which they are added. If the fillers are characterized with a low aspect ratio between the longest and the shortest dimensions, the basic properties will be less changed from those of the unfilled polymer. Fillers benefit plastics parts in the following ways:

- Shrinkage will be less.
- Thermal resistance may be improved.
- Strength, especially compressive strength, will be improved.
- Impact resistance will often be lower than for the unfilled polymer.
- Solvent resistance will often be improved.

D.5.24 High-aspect Fillers: Fibers

When the aspect ratio between the longest and the shortest dimension of the filler is large (for example, greater than 25) the filler can be characterized as a fiber. Fiber reinforcements will significantly affect the properties of the compounds to which they are added.

D.5.24.1 Fibers Impact Strength

Assuming good bonding between the fiber and the polymer matrix, the strength in the fiber direction will be significantly increased. If many fibers are oriented in the same direction, large differences will be noted between the modulus in the orientation direction and in the direction perpendicular to the orientation. The latter will be very close to that for the unfilled polymer.

D.5.24.2 Fibers Affect Shrinkage

The fibers will also have a significant effect on the shrinkage properties of the compound: shrinkage in the orientation direction will be much less than the shrinkage in the cross direction.

D.5.24.3 Importance of Predicting Fiber Orientation

Because the fiber orientation varies with the flow direction, in the thickness direction, and at weld line locations, it is important to be able to predict these orientations to determine the properties of the molded article.

Index

Numerics

3D mesh 62

A

ABS 290

Additives. See Plastics

Air traps 64, 65, 122, 215, 221

eliminating 216

location of, 215

prediction 65

problems caused by 216

Air-shot temperature 272

Amorphous polymers 184, 286

Analysis

objectives 191

sequence 47

steps 192

Area shrinkage 181, 182

Assembly

annular snap-fit joints 94

cantilever snap joints 95

designing for 84, 90

fasteners 98

press-fit joints 91

snap-fit joints 93

torsion snap-fit joints 96

welding processes 101

Assumptions 63

B

Back pressure 274

Backflow 26

Baffles 157

Balanced

filling 116

Black specks and streaks 217

Brittleness 219

Bubblers 157, 159

Burn marks 220

C

Chisel gate 111

Clamp tonnage capacity 137

Coefficient of thermal expansion 82

Co-injection molding 249

Cold runners 127, 134

Coolant flow 163

Cooling

channels 156

cores 160

optimization 49, 192, 199

system 154, 207

time 30, 162, 275

uniform 54

Cooling channel

flow rate 166

location 164

size 164

Crystalline polymers 184

Crystallinity 178

Cycle time 86, 153

D

Delamination 222

Diaphragm gate 108

Differential cooling 182

Dimensional variation 224

Discoloration 225

Disk gate 108

E

Edge gate 106

Elements 61, 63

F

Fan gate 107

Fatigue 79

Feed system 127

See also Runners

Filling 174

analysis 206

optimization 48, 192

pattern 33

Film gate 109

Finite element 61, 63

Fish eyes 227

Flash 228

Flash gate 109

Flow

balancing 51

concepts 49

effect of temperature and pressure 11

front area 41

front velocity 41

hesitation, avoid 55

in complex molds 34

leaders 57

length 13, 67

multi directional 39

pattern 41, 48, 115

pressure gradient 13

unidirectional 48

unstable 40, 176

Flow concepts 49

acceptable runner/cavity ratio 58

avoid hesitation effects 55

avoid underflow 55

balance with flow leaders and flow

deflectors 57

constant pressure gradient 52, 133

controlled frictional heat 58, 132

flow balancing 51

positioning weld and meld lines 55

shear stress, maximum 53

thermal shutoff of runners 58, 132

unidirectional filling pattern 50

uniform cooling 54

Flow Marks 230

Frictional heat 58

Fusible core injection molding 250

Fusion mesh 62

G

Gas assisted injection molding 251

Gates 103

air traps 122

automatically trimmed 110

chisel 111

design rules 113

determine the number of 48, 103, 113,
194

diaphragm 108

dimensions 103

disk 108

edge 106

fan 107

film 109

flash 109

flow pattern 115

freeze time 123

hot 112

hot-drop 112

hot-probe 112

hot-runner 112

jetting 122

length 122

location 103, 210

manually trimmed 104

overlap 107

overpacking 121

pin 111
positioning 48, 116, 185
pressure 120
ring 109
shear rate 122, 125, 206
shrinkage 175
spider 109
spoke 109
sprue 105
standard 106
sub 111
tab 106
thickness 122
tunnel 111
type 104
valve 112
wall thickness 118
Geometry 69

H

HDPE 301
Heat deflection temperature 82
Heated runners 135
Hesitation effect 37, 64, 116, 231
Hot drop 112, 128
Hot gate 112
Hot manifold 128
Hot runners 128, 135
Hydraulic diameter 144

I

Impact strength 80
Injection molding 248
 compensation phase 6
 filling phases 3
 fusible core 250
 gas assisted 251
 injection compression 252

lamellar 253
live-feed 253
low-pressure 254
machine 195, 259
operating sequence 266
overview 247
phases 1
 pressurization phase 6
push-pull 254
structural foam 256
thin-walls 256

Injection pressure 17
 See also Pressure, injection
Injection rate 35
Insulated runners 135

J

Jetting 116, 122, 232

L

Lamellar injection molding 253
LDPE 303
Live-feed injection molding 253
Low pressure injection molding 254

M

Machine 248
 operating sequence 266
Meld lines 38, 55, 243
Melt
 shear viscosity 9
 temperature 272

Mesh

3D 62
density 64
Fusion 62, 63
midplane 62, 63
requirements 64
tetrahedral 62

- Mesh types 62
- Midplane mesh 62
- Mold 263
 - Three-plate 263
 - Two-plate 263
- Mold temperature 208, 272
- Molding conditions 21, 48, 195, 271
 - fill time 23
 - melt temperature 22
 - mold temperature 22
 - molding window 27, 205
 - shear stress 23
- Molding machine 195
- Molecular orientation 180
- N**
- Non-Newtonian 10
- Nylon 291, 293, 294
- O**
- Orientation 180, 181, 182
- Overlap gate 107
- Overpacking 34
- P**
- PA 12 291
- PA 6 293
- PA 66 294
- Pack time 201, 275
- Packing 174, 192, 200, 209
- Parallel cooling channels 156
- Part
 - design 67
 - design, See also Product Design
 - details 67
 - flow length 67
 - geometry 69
 - thickness 67, 86
- Part quality 21
- PBT 296
- PC 297
- PC/ABS 299
- PC/PBT 300
- PEI 304
- PET 305
- PETG 306
- Pin gate 111
- Plastic
 - deformation 8
 - flow 7
 - non-Newtonian 10
 - shear viscosity 9
 - shear-thinning 10
 - viscoelastic behavior 8, 73
- Plasticizing capacity 137
- Plastics 281
 - additives, fillers, reinforcements 317
 - amorphous 286
 - classes 284
 - engineering resins 286
 - semi-crystalline 287
 - structures and properties 284
 - thermoplastics 285, 289
- PMMA 307
- Polycarbonate 297
- Polymer 281
 - alloys and blends 283
 - composites 283
- Polymerization 281
- POM 309
- PP 310
- PPE/PPO 312
- Pressure
 - gradient 52, 133
 - injection 17, 28
 - injection, factors that influence 18
 - injection, vs fill time 13

- packing 24
 - vs wall thickness and flow length 68
- Process conditions 271
- Processing conditions
 - See Molding conditions
- Product design 47
 - creep 77
 - fatigue 79
 - heat deflection temperature 82
 - high velocity and impact loading 85
 - impact strength 80
 - loading at extreme temperatures 85
 - long-term loading 83
 - material properties 71
 - part strength 72
 - repeated loading 84
 - ribs 88
 - short-term loading 83
 - strength, design for 82
 - stress relaxation 79
 - stress-strain 72, 75, 76, 77
 - tensile properties 72
 - thermal mechanical behavior 80
 - thickness 86
 - viscoelastic behavior 73
 - young's modulus 75
- PS 313
- Push-pull injection molding 254
- PVC 314
- PVT 177
- R**
- Racetrack effect 35
- Reciprocating screw 268
- Reynolds number 163
- Ring gate 109
- Ripples 235
- Runners
 - artificially balanced 139
 - balance 51
 - balanced 138
 - balancing 130, 147, 192, 197
 - cold 134
 - cold slug well 133
 - cross-sections 142
 - design 48
 - design principles 129
 - diameters 145
 - flow control 130
 - freeze time 148
 - freezing 58
 - frictional heating 132
 - geometrically balanced 141
 - heated 135
 - hot 135
 - hydraulic diameter 144
 - insulated 135
 - layout 136, 138
 - linear shrinkage 148
 - partially balanced 139
 - pressure 132, 147
 - pressure gradient 133
 - processing window 148
 - runner/cavity ratio 58
 - sizing 141
 - sprue sizing 141
 - temperature 148
 - thermal shutoff 132
 - types 134
 - unbalanced 138
 - volumetric shrinkage 148
- S**
- SAN 316
- Sandwich molding 249
- Screw 268

- Screw rotation 274
 - Semi-crystalline polymers 287
 - Series cooling channels 157
 - Shear heat 58
 - Shear rate 11
 - Shear stress 4, 23, 53
 - Shear-thinning 10
 - Short shot 236
 - Shot capacity 137
 - Shrinkage 173
 - area 181
 - causes 177
 - crystallinity 178
 - end of flow 176
 - gate 175
 - linear 178
 - pressure 175
 - volumetric 177, 178
 - Silver streaks 238
 - Sink marks 38, 240
 - Solver assumptions 63
 - Specks 217
 - Spider gate 109
 - Spoke gate 109
 - Sprue 128
 - sizing 141
 - Sprue gate 105
 - Streaks 217
 - Stress relaxation 179
 - Stress-strain 72, 75, 76, 77
 - Structural foam injection molding 256
 - Sub gate 111
 - Switch-over 273
- T**
- Tab gate 106
 - Temperature
 - flow front 29
 - instability 7, 176
 - variation 7
 - Tensile properties 72
 - Thermal mechanical behavior 80
 - Thermal pin 157, 159
 - Thermoplastics 285
 - Thermoplastics properties 289
 - Thermosets 288
 - Thin-wall molding 256
 - Tunnel gate 111
- U**
- Underflow 36, 48, 55, 115
- V**
- Valve gate 112
 - Vents 122
 - Voids 240
 - Volumetric shrinkage 177, 178
- W**
- Wall thickness 67
 - Warpage 173
 - analysis 209
 - area shrinkage 182
 - causes 181, 203, 210
 - differential cooling 182
 - optimization 49, 192, 201
 - orientation 182
 - orientation effects 181
 - reducing 204
 - wall thickness 184
 - Weld lines 38, 55, 116, 243
 - prediction 65
- Y**
- Young's modulus 75