English

Polypropylene

Injection Molding Polypropylene

basell
Polyolefins

Contents

Injection Molding Polypropylene

About Basell

Basell develops, produces and markets polypropylene, polyethylene, advanced polyolefin materials and polyolefin catalysts and also develops and licenses polyolefin processes.

Formed in October 2000, Basell is owned equally by BASF and Shell. Basell and its joint ventures serve customers in more than 120 countries with materials produced in 18 countries. The company's network of joint ventures expand Basell's technology and market base and enable the company to follow key customers as they expand and globalize their operations.

With research and development centers in Europe, North America and the Asia-Pacific region, Basell is continuing and expanding a technological heritage that dates back to the start of the polyolefins industry. The company is committed to continuously extending the property profile of its polyolefins portfolio and to developing with its customers a shared agenda for bringing new products to market as quickly as possible.

Basell is committed to strong Health, Safety and Environmental (HSE) performance. The company's products are used in countless consumer and industrial goods from food and drink packaging to car components, and from household products to underground piping.

Basell's corporate centre is located in Hoofddorp, The Netherlands, near Amsterdam. The company has regional offices in Brussels, Belgium; Mainz, Germany; Wilmington, Delaware, USA; Sao Paulo, Brazil and Hong Kong, as well as sales offices in the major markets around the world.

Introduction

This technical guide should provide molders with general insight into the characteristics of unfilled or natural polypropylene and aspects of design and processing for optimum results. The guide also focuses on the general procedures for correcting common molding problems.

Prospective molders may wish to consult their Basell technical representative to choose the right polypropylene grade for their application or to obtain further assistance with product design and processing.

Characteristics of polypropylene

Like all thermoplastic injection molding resins, polypropylene has its own special characteristics. These characteristics not only affect the properties of the finished molded pieces, but they also determine optimum molding conditions. Available as homopolymer, random copolymer or impact copolymer types, polypropylene is offered in a broad range of grades and types which have properties that are fully outlined in their respective product data sheets.

Where we refer to polypropylene in this guide, we include both Basell's homopolymers and copolymers sold under the trademarks *Pro-fax* and *Moplen*. All products made using our *Spheripol* process would be expected to behave similarly. Although minor differences in flow, shrinkage, cooling and hinge life may exist, these differences are small.

I. Inherent physical and chemical properties

Low density

All types of natural or unfilled polypropylene have the same very low density of 0.90 g/cm³ that is the lowest of all commonly available thermoplastics. Parts molded from polypropylene are lighter weight, and therefore more parts can be molded on a partper-weight basis.

High-temperature resistance

The relatively high melting point of 334℉ (167℃) for Pro-fax and Moplen polypropylene allows continued use at 220℉ (104℃). The resin begins to soften at about 250℉ (121℃), but nevertheless can be used intermittently at this temperature. To extend polypropylene's useful temperature range and service life, an antioxidant system is incorporated. However, any environment (such as moisture) that tends to extract the antioxidants may lead to a more rapid breakdown of polypropylene, especially at elevated temperatures.

Chemical resistance

Polypropylene, like most polyolefins, is highly resistant to solvents and chemicals. With few exceptions, inorganic chemicals produce little or no effect on polypropylene exposed to temperatures up to 250℉ (121℃) for a six-month period. Polypropylene is quite resistant to polar organic chemicals but is subject to swelling and softening by non-polar solvents, such as benzene, toluene, carbon tetrachloride, etc. Suitability for use in these environments should be determined by testing. Compatibility data with common chemicals are available.

Stress-crack resistance

Polypropylene has excellent resistance to environmental stress-cracking. Embrittlement that occurs with other plastics in the presence of oils, detergents, and other stress-cracking agents is not observed with this resin. Generally, only very potent oxidizing agents produce stress-cracking in polypropylene.

Weathering (ultraviolet resistance)

Polypropylene has limited resistance to weathering or exposure to ultraviolet light, a component of sunlight. The incorporation of 2.0 to 2.5% carbon black pigment has protected Basell polypropylene parts outdoors for up to 20 years. If black is not an acceptable color, incorporation of an ultraviolet stabilizer should be considered.

Radiation sterilization

Polypropylene resins, in general, are not recommended for applications where radiation sterilization is required due to both yellowing and loss of physical properties including embrittlement. Basell offers several grades that are specifically formulated to minimize the effects after typical radiation sterilization dosages of up to 5 Mrad, as tested by Basell protocol. Basell recommends that prospective users conduct their own tests to determine the suitability for the intended purpose.

Heavy metals

Basell natural (unfilled) polypropylene polymers have been tested for the following heavy metals, and these metals were not found at the sensitivities listed in parentheses: antimony (3 ppm), arsenic (2 ppm), barium (2 ppm), cadmium (1 ppm), chromium (1 ppm), lead (2 ppm), mercury (0.01 ppm), selenium (3 ppm), and silver (1 ppm).

Other advantages

- Excellent dielectric properties
- Non-hygroscopic, does not absorb moisture or require drying
- Excellent dimensional stability
- High gloss or matte, based on mold surface finish
- Best contact and see-through clarity of the polyolefins

These optical properties can be further enhanced by the addition of nucleating agents, which form smaller and more numerous spherulites upon crystallization. Furthermore, polypropylene grades containing nucleating agents increase crystallization rates which change the processing behavior as well as modifying the stiffness and impact properties of the molded part.

II. Mechanical properties

Polypropylene has excellent mechanical properties. The numerous homopolymer and copolymer grades offer various combinations of stiffness and impact strength to meet the specific requirements of many injection molding applications.

Stiffness

Stiffness is defined by the measurement of the flexural modulus on a molded specimen. Of the polypropylene family, homopolymers possess higher stiffness than both the random and impact copolymer varieties.

Polypropylene resins are intermediate in stiffness to that of polystyrene and high density polyethylene (HDPE). High impact copolymers and random copolymers are similar to the flexural modulus (stiffness) of HDPE, while homopolymer polypropylenes can be stiffer than impact modified polystyrene.

Impact strength

The impact strength of polypropylene can be measured in several ways. The most common methods of measurement are the impact strength as determined by a pendulum type apparatus striking a notched specimen (Izod and Charpy) and the drop weight impact strength as determined by a failing weight on a molded specimen. The impact strength reported is greatly dependent on test temperature.

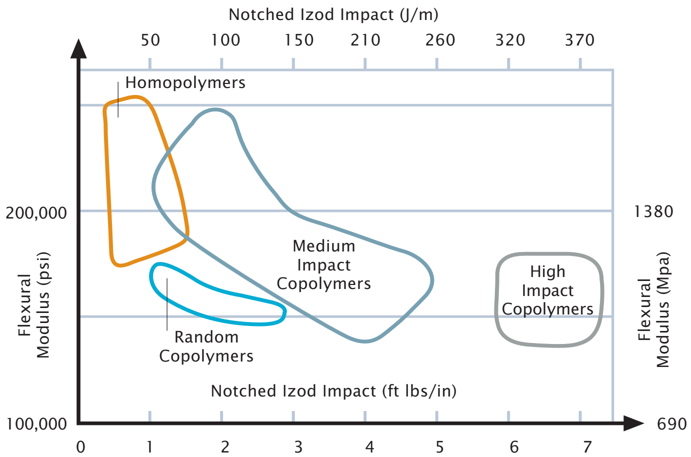


Fig. 1 Impact strength properties

For many applications, *Pro-fax* and *Moplen* polypropylene homopolymer provide adequate impact strength at or above room temperature. However, for applications with requirements for low temperature impact resistance, impact copolymers are recommended. These grades not only improve the impact properties but also reduce the brittleness temperatures of molded parts.

Like other crystalline thermoplastics, polypropylene is relatively notch sensitive. Thus, notched areas such as sharp comers require a radius to minimize stress and improve impact resistance in molded articles. The recommended corner design is shown in Figure 8.

III. Long-term creep

The end use performance of injection molded articles made from polypropylene are almost always related to stress, time and temperature.

The ability of a product to resist deformation under a constant load applied over time is the measure of creep resistance. Tensile creep is defined as deformation in tension, rated in percentage strain that occurs under applied stress.

Polypropylene has tensile creep resistant properties superior to those of other polyolefins, and these properties can be further improved by the addition of glass fibers and other modifiers. Separate data are available on these products.

Figures 2 and 3 depict the tensile creep data for general purpose *Pro-fax* and *Moplen* polypropylene homopolymer and copolymer. As shown, the homopolymer exhibits better creep resistance than does the copolymer. Different products will show different creep behavior and can be typically related to resin stiffness (flexural modulus).

Furthermore, consideration should be given to design and molding conditions, as they may impart "molded-in" strain and orientation effects. These molding-induced effects will affect the behavior of the molded parts.

IV. Flow properties

From a material standpoint, the moldability of polypropylene is determined primarily by its flow properties. Of importance are the following tests:

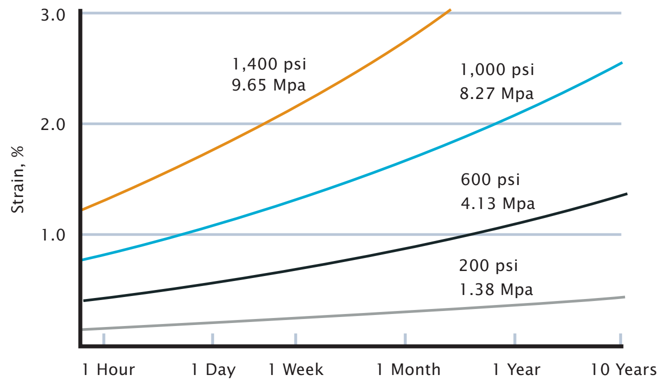


Fig. 2 Typical *Pro-fax* 6623 polypropylene homopolymer tensile creep at 70℉ (21℃)

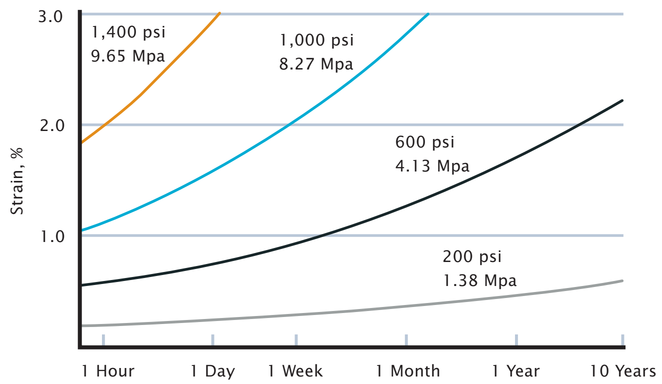


Fig. 3 Typical *Pro-fax* 7523 polypropylene copolymer tensile creep at 70° (21℃)

Melt flow range, g/10 min	Flow characteristics
Below 4	Low flows
4 to 10	Medium flows
10 to 20	High flows
20 to 40	Extra high flows
40 to 100	Ultra high flows

Fig. 4 Simplified rating of the available melt flow ranges into flow characteristics:

Melt flow rate

The melt flow test method, according to ASTM D 1238, (ISO 1033) is a measurement of the rate of extrusion (reported in grams/10 minutes) of the molten resin through an orifice of a specified length and diameter under prescribed conditions of temperature and pressure. The melt flow rate (MFR) used for polypropylene should not be confused with the melt index (MI) for polyethylene. The apparatus is the same, but the melt flow rate is determined at 446℉ (230℃), 4.76 lbs. (2.16kg) piston force, while the melt index of polyethylene is at 374℉ (190℃), 4.76 lbs. (2.16kg) piston force.

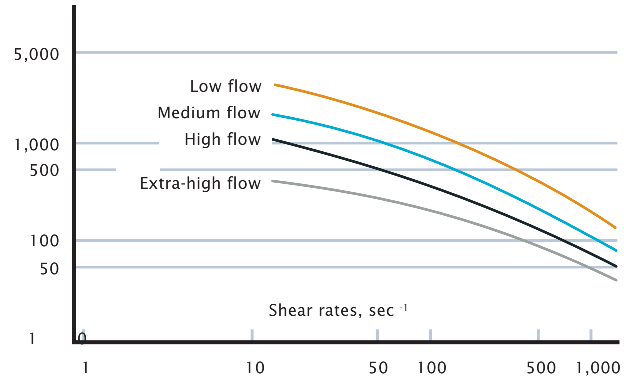


Fig. 5 Typical flow characteristics at 446℉ (229℃)

The melt flow rate is a single point measurement of the melt viscosity (melt's resistance to flow) and is also used to give an estimate of polypropylene's molecular weight. The melt flow rate is inversely related to the molecular weight (i.e., as the molecular weight increases the melt viscosity increases and the melt flow rate decreases).

At the low shear rates employed for the test method, the melt flow rate cannot be used by itself to give a reliable measure of moldability, owing to the high shear rates encountered in injection molding.

Despite this limitation and the limited precision of the melt flow results attributable to the procedure itself, the melt flow apparatus is nevertheless widely used in the plastics industry as an economical, convenient and fast testing device. This enables users to distinguish the variety of available injection molding grades intended to meet their specific needs.

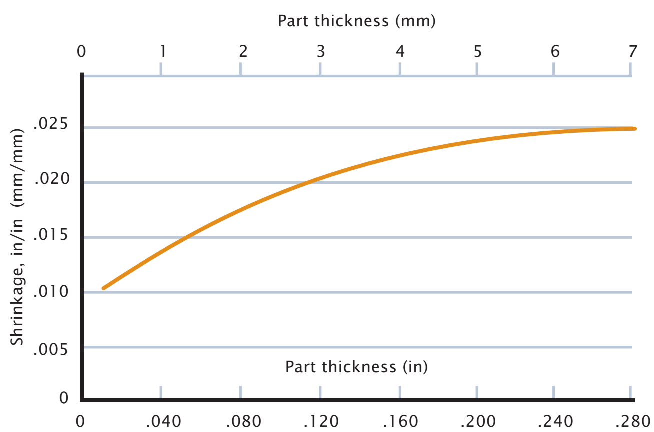


Fig. 6 Pro-fax polypropylene shrinkage variation with thickness

Note: Shrinkage values given here are the maximum to be expected under most molding conditions (primarily based on cooling rate). Although the data are useful in determining mold cavity allowances, it should be remembered that lower values can be obtained when molding conditions are optimized by adjustment of cycle times, temperatures and other process variables.

Capillary flow

The capillary flow test, using a capillary rheometer, measures the viscosity of the molten resin at high shear rates similar to those that occur in the injection molding process. This technique provides a better assessment of the resin's moldability.

Above is an illustration of the viscosity vs. shear rate curves of typical flow characteristics of polypropylene.

Polypropylene, when subjected to high shear rates, will experience a dramatic decrease in viscosity that aids in mold filling. This shear thinning property can be particularly useful for long flow paths on large molded parts. On the other hand, the polymer is subjected to very high stresses that may affect the final part properties, so care must be exercised in using excessively high shear rates.

Designing parts for polypropylene

I. Part design considerations

Wall thickness - effect on shrinkage

The wall thickness of a molded part has a direct effect on polypropylene shrinkage. This contraction of the part occurs predominantly within the first 48 hours after molding due to polymer post-melt crystallization.

The rate of cooling (related to melt temperature, mold temperature and mold closed time) as well as second stage packing pressure directly influences the extent of shrinkage. Thick sections naturally cool slower and therefore shrink more as depicted in the graph below. This relationship favors the design of molded parts with a constant wall thickness; otherwise internal stresses may cause warpage. Mold filling/packing can be a major cause for inconsistent shrinkage values. Walls should be less than 1/4" (6.4mm) thick to avoid shrink voids especially for areas far from the gate that may be difficult to pack out.

Thin wall molded parts of 0.020 inch (0.5 mm) or less can have shrinkage values of 0.020 inch/inch (0.020 mm/mm) or more based on molding conditions. Prototype molds are advised for new applications where precise final part dimensions are required.

As shown in Figure 7, if a part were designed with double the necessary wall thickness, the cooling time (which is also sensitive to melt temperature, mold temperature, resin stiffness and the rate of crystallization) would need to be increased 3 to 4 times.

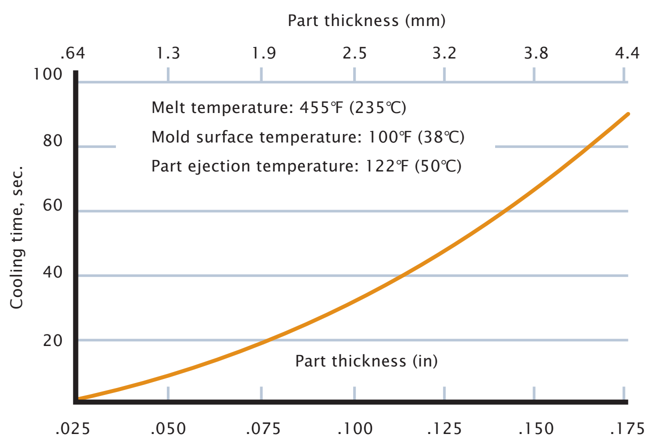


Fig. 7 Cooling time vs. part thickness

- Nucleation

Polypropylene can be nucleated to produce dimensionally stable parts. Most of the post molding shrinkage occurs in nucleated parts in the mold. The mold, therefore, becomes a cooling jig stabilizing the part to the mold shape, and preventing warpage. However, the parts can be difficult to strip owing to shrinkage on cores. Nucleated parts also tend to freeze in molding stresses and parts may possess slightly less impact resistance than similar parts made with non-nucleated polypropylene. Certain polypropylene resins, particularly random copolymers, have clarifiers added to them that also behave as nucleators. Be aware that certain pigments are excellent nucleators (e.g. green blue, orange, etc.) such that molded parts of different colors may result in different part dimensions.

- Flow consideration

To determine the minimum wall thickness from a moldability standpoint, the flow of the resin in the mold should be considered with respect to melt and mold temperatures, cavity length and runner and gate sizes.

In general, thin-walled parts with long flow paths require extra or ultra high flows, while thick-walled parts with short flow paths allow the use of medium to low flow resins. Molded parts with living hinges frequently require higher flows to ensure a rapid mold fill and good hinge quality. Too high a melt flow reduces the flex life of the hinge.

■ Guidelines

Wall thickness should be uniform wherever possible to minimize warpage and sink marks. If this is not possible, wall thickness should decrease progressively in the flow direction.

Ribs

Ribs are intended to stiffen and strengthen the molded part while maintaining a minimum wall thickness. Furthermore, they help control the resin's flow in the cavity and prevent warpage of critical areas in the part.

The base of the rib should have a radius of about 25 to 50% of the nominal wall thickness. Studies have shown that stress (and therefore notch sensitivity) is minimized with a radius equal to 50% of the wall thickness.

The base of the rib should have a width approximately 75% of the adjacent nominal wall thickness. The rib should further be tapered 0.5 to 1° per side, and should have a depth of 1.5 times the nominal wall. Deeper ribs can be used; however, the usually require a thicker base to allow for the draft angle. Ejection problems may increase when deeper ribs are used and can result in sink marks at the wall intersection under the base of the rib.

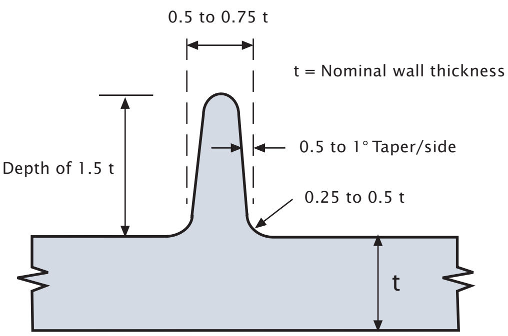


Fig. 8 Recommended rib design

As shown above, a properly designed rib is thin and shallow, rather than thick and deep

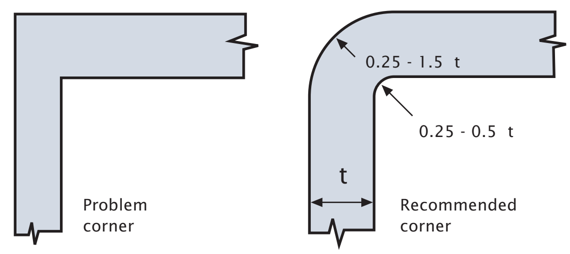
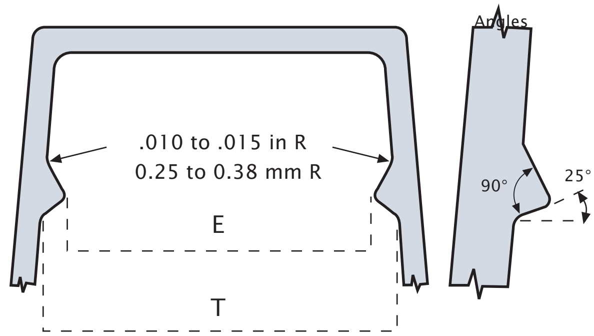


Fig. 9 Correct wall thickness at corners

Radii

Correctly designed parts should include radii to distribute stress and reduce the relative notch sensitivity of polypropylene. They should be included at all sharp comers, inside and out.

The inside corner radii should be 0.25 to 0.5 times the wall thickness. The outside comer radii should be 1.25 to 1.5 times the nominal wall thickness. Applying these guidelines helps to maintain a constant wall thickness.



Undercut size (%) or strain (deformation) rate (%) = T – E x 100

Fig. 10 Diagram of an internal undercut

Draft

To improve part ejection, draft angles should be provided on the inside and outside of part walls in the direction of draw. A draft of 10 per side is adequate, although a larger angle will make it easier for part release. The use of a nucleated polypropylene may require more draft as part shrinkage occurs primarily on the mold core versus more post-mold shrinkage than when using a similar non-nucleated polypropylene. Though less than a 10 draft angle may be used in certain instances, its use should be based on molding experience with similar parts. Textured surfaces require an additional draft of 10 per side for each 0.001 in (0.025 mm) of texture depth.

Undercuts

Undercuts should be avoided. However, for articles (such as closures) that require threaded undercuts or snap fits, the undercuts should be designed with a lead-in angle of approximately 250 to facilitate parts stripping. The undercut root should also incorporate a radius of 0.010 to 0.015 in. (0.25 to 0.375 mm) or more, where possible.

The size of the undercut for a circular part (Fig.10) is determined as the percentage difference between the outer diameter “T”, and the inner diameter “E.” The percentage of undercut size represents the allowable percentage rate of deformation (strain) of the undercut. It is Basell’s experience that undercuts which cause more than 5% deformation can permanently deform the part.

Living hinges

Living hinges using unfilled or natural polypropylene achieve outstanding service life. Basell has published a separate guide to cover design and molding requirements for living hinges. Please consult the guide on Living Hinges for further information.

Bosses

Bosses are typically used as mounting fixtures. They can be compare to a rib in as much as the requirements for draft angle and comer radiuses. To reduce the possibility of sink marks, careful design is also required to maintain a constant wall thickness. The figures to the right depict different boss design approaches. They show the interior hole should be extended to within % of the nominal wall thickness to minimize sink marks.

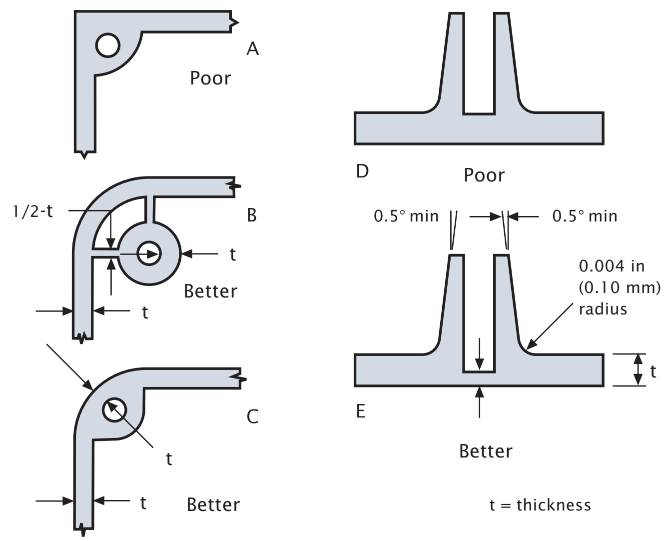


Fig. 11 Recommended boss design

Holes

Where a hole is required in a molded part, the design must take into consideration conditions to minimize stress. Holes should be positioned at least a distance equivalent to the hole’s diameter and no more than twice the hole’s diameter away from the side wall. Consideration should also be given to minimize weld line formation with a careful review of gate location as the hole will restrict flow. As with bosses and ribs, all comers should be radiused.

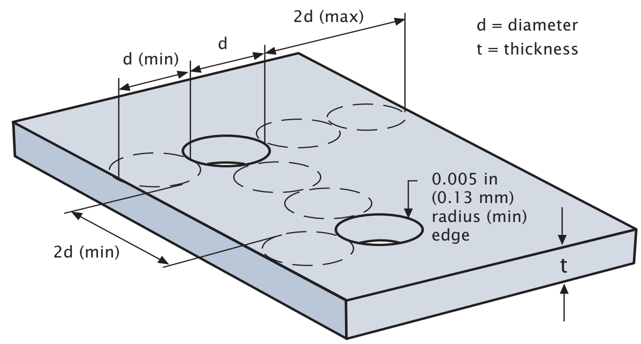


Fig. 12 Recommended hole design

II. Mold design considerations

Design of gates

Gate design is a major decision in mold construction. Gate location, size, and type will influence ease of molding, part dimensional stability, toughness, appearance and the need for trimming. Problems relating to venting, core deflection and "reweld" (weldlines) can be prevented or solved using the proper gate and location.

Basic gating requirements include:

- Must remain open until part is filled and packed
- Must allow continuous, uninterrupted polymer flow in the cavity
- Must allow part to be filled under reasonable pressure and temperature without polymer degradation, pigment separation or color changes (where part is pigmented)

The subgate (or tunnel gate) is a very popular design used in the production of polypropylene parts as parts are automatically degated when the mold opens. The subgate is fed by a parting line runner system, which does not intersect at the parting line but tunnels to a location on the part on the moving side of the mold. The gate is trimmed off by the ejection action, aided by the acute angle in the mold steel which acts as a blade.

Gate designs that do not intersect the part at the parting line are the sprue gate, the pin gate (a type of full round) and the subgate (or tunnel gate). The sprue gate leaves a relatively large round gate vestige tapering away from the part, which must be snipped off. If appearance is important, it can be machined away. Pin gates are used with both three-plate molds and hot runner systems. These types normally avoid trimming or degating operations.

A number of gate types are located at the parting line. The usual types are: (1) flat or tab gate which is a simple rectangular channel commonly used because it is easy to machine and size accurately; (2) fan gate designed so that the flow diverges to a wide, shallow intersection with the part; (3) flash gate where a thin membrane is formed between a

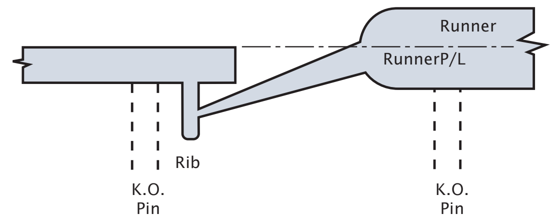


Fig. 13 Subgate

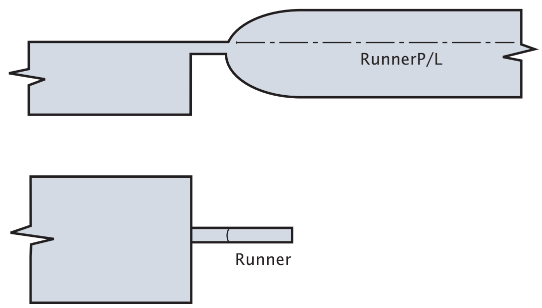


Fig. 14 Tab gate

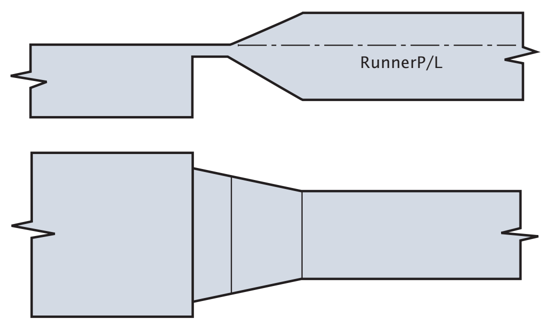


Fig. 15 Fan gate

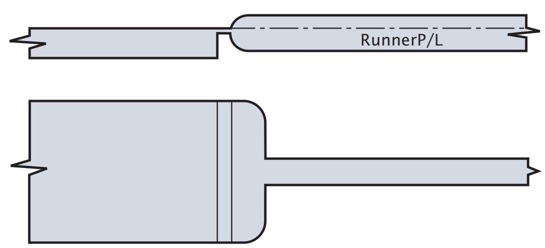


Fig. 16 Flash gate

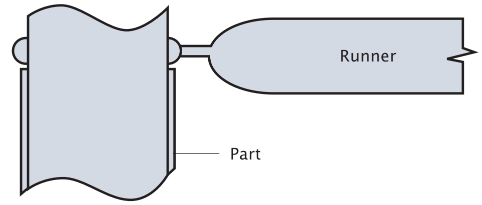


Fig. 17 Ring gate (a type of flash gate)

runner and a long side of the part which is normally used in parts which cannot be filled from a single point; (4) ring gate which is a modification of a flash gate that would surround a tubular part; and (5) diaphragm gate which is the opposite of a flash gate in that it feeds the interior of a tubular part. In general, parting line gates require mechanical trimming.

Gate sizing is based on part design. A rule of thumb is half the nominal wall thickness with a 0.035 inch (0.090 mm) minimum diameter. Only parts weighing less than an ounce (30 g) can readily be filled with a 0.025 inch (0.065 mm) diameter gate. Hot drops form a melt annulus as most designs have a heated core in the center of the runner. Therefore, a 0.035 inch (0.90 mm) diameter hot drop produces a 0.015 inch (0.38 mm) thick annular ring vestige on the molded part.

For similar performance, gates must have identical land lengths (the parallel wall section as the gate meets the cavity) as well as gate areas. Land lengths for polypropylene should be 0.015 to 0.020 inch (0.38 to 0.50 mm). Gates without lands cannot be dimensioned, dull quickly and may cause the mold steel to crack. Tiny changes in small gates produce large changes in performance; therefore, balancing of multiple cavities is best done with the runner system.

The melt flow rate of the resin used, the runner design, the gate configuration and the length of flow in the part are major considerations in gating. Thin gates, such as the flash gate, freeze off very promptly after injection, while sprue gates remain open for many seconds.

Gate location

Because the gate area is often highly stressed, it should be located so that the product's properties and appearance are not adversely affected.

Gate location should:

- Ensure a balanced flow (rapid and uniform filling) in the cavity so that no areas of the part are overpacked
- Gate in the thickest section and direct material flow from thick to thin sections whenever possible
- Ensure mold fill under realistic temperatures and pressures
- Minimize weldlines as much as possible or position them in non-critical areas
- Prevent "jetting" by positioning the gate so that material flow is smooth and uniform
- Avoid air entrapment
- Avoid visual distraction from overall part appearance

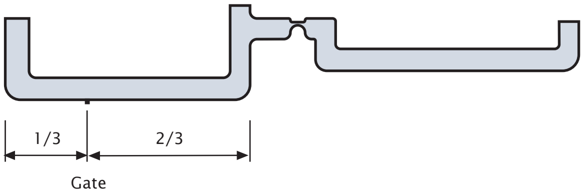


Fig. 18 Correct gate location for shallow boxes

For parts with an integral hinge, the gate should be located beyond the centerline of the cavity, opposite the hinge (not parallel to it) and well away from it so as to allow the hinge to be the last portion of the cavity to fill. This is particularly true for the molding of shallow boxes where the most suitable gate should be located as shown right:

Vent location

All cavities must be sufficiently vented so that the air displaced by the incoming material flow is properly evacuated. Insufficient vents can cause bum marks, bubbles, slow and incomplete fill and weak weldlines.

Vents should be located as follows:

- In the last section of the mold to be filled
- Where flow fronts meet and form weldlines
- Near projections or blind spots, such as ribs and bosses
- Vents for polypropylene should be sized as follows:
  - Depth: Approximately 0.001 to 0.002 in (0.025 to 0.06 mm); 0.0015 in (0.038 mm) is typical
  - Land lengths: 0.015 to O.040 in (0.38 to 1.0 mm)
  - Width: As wide as possible
- A minimum of 30% of the parting line surface should be vented

Cooling requirements

For faster cycles, mold cooling requirements must be considered from the start. The cooling system should balance the heat flow from the part to ensure uniform part cooling and minimize residual stresses, differential shrinkage and warpage. Cooling the mold below the dew point should be avoided because it will cause condensation and molding problems.

Suggested cooling channel guidelines are as follows:

- A cooling channel diameter of 7/16 to 9/16 inch (11 to 14 mm)
- A cooling channel whose distance from the surface is 1 to 2 times its diameter
- A pitch (distance between the cooling channels) of 3 to 5 times its diameter (the gate area is the warmest part of the mold, therefore the pitch should increase toward the vent section of the mold)

Should improvement in cooling be required in an existing mold, the following steps are recommended:

- Increase the coolant flow rate (primary reason for poor cooling)
- Reduce the coolant temperature (most expensive alternative)
- Clean out the buildup of sediment in the channels
- As a recommendation for chiller capacity, allow 1 ton of capacity for every 35 lbs. (16 kg) of polypropylene to be processed per hour

Ejection systems requirements

Parts made with polypropylene release readily from molds. A properly designed ejector system will facilitate part removal as long as the following design principles are adhered to:

- A matte mold core surface (grit blasted or draw polished) facilitates stripping
- Ejector pins must be strong enough to withstand forces encountered in part ejection
- Ejector pins should be positioned to evenly distribute the load across the part
- Minimum clearances between pins and mold should be used to eliminate flashing around the pins
- Ejector pin location should be consistent with providing aesthetically pleasing part surface finish
- Ejector pins should not be located near part wall intersections to allow cooling channels to be located where they will be most effective

Ejection of molded parts is not limited to the use of ejector pins. The use of a stripper plate is another common ejection system for circular parts or where the ejection force is evenly distributed across the part surface.

Multicavity mold design

If a multicavity mold will be used vs. a single cavity tool (based on machine size, part quantity, delivery constraints and molding cost), careful consideration should be given to the balancing of the mold runner system.

Balance is accomplished by a runner design that locates all parts equidistant from the sprue and reducing the runner volume by varying the area and shape of the runners and hot runners. If possible, do not balance by varying the gate sizes.

Ultimately, the ideally balanced mold should deliver polypropylene to each gate under identical conditions (i.e., temperature and pressure). Unbalanced molds usually cause cavities to fill at different pressures and speeds resulting in overpacking, warpage, differential shrinkage, brittleness, dimensional inconsistencies and parts that stick to the mold. A "short shot" evaluation will provide the molder with a reasonable assessment of fill balance.

Molding parts with *Pro-fax* and *Moplen* polypropylene

I. Injection machine requirements

Rated capacity

An estimate should be made of the shot size required for the particular molded part. This estimate should be compared with the machine's rated capacity. Polypropylene shot sizes have a maximum limit of approximately 75% of the machine's rated capacity in polystyrene. When using color concentrates, the shot sizes have an approximate maximum limit of 60% to allow for adequate mixing and dispersion.

Screw type

A typical screw used for polypropylene is the single-stage, general purpose type with a length-to-diameter (L/D) ratio of 16:1 to 24:1 and a compression ratio between 2.5:1 and 3.0: 1. A two-stage screw with a vented barrel is not required.

Clamping force

The clamping force required for polypropylene is 2 tons/in 2 (2.8 kg/mm^2) of projected area for a wall thickness greater than or equal to 0.090 in. (2.3 mm). Thinner walls require slightly more clamp tonnage. Therefore, an overall average clamp tonnage of 2.5 tons/in 2 (3.5 kg/mm^2) is commonly recommended for parts with wall thicknesses of less than 0.090 in. (2.3 mm).

II. Startup molding conditions

Polypropylene has been successfully processed on a wide variety of injection molding machines, and general assumptions have been made to simplify the following recommendations. It is assumed that the machine is fitted with a reciprocating screw injection unit with the screw having a minimum 20:1 L/D ratio and a compression ratio of at least 2.5:1. It is assumed that the mold is in reasonably good condition (e.g. not flashing, vents not hobbed shut, etc.).

On a cold start up, allow sufficient time at initial set conditions to make sure that all parts (including the screw) have reached processing temperatures. Resins with different melt flows will require different settings as shown below. Barrel and nozzle temperatures may be the same (flat profile) although some molders prefer to progressively increase temperatures from the hopper to the nozzle. If a hot manifold is being used, the same temperatures would apply.

Run a few “air shots” to check melt quality (no unmelted pellets or excessive smoking). Mold cooling can be left off during the startup to prevent sweating. It is usually not necessary to purge the machine of previous material as polypropylene is commonly used as a purge resin.

Advance the injection unit, engage the nozzle in the sprue bushing and run a few shots while checking for shorts and flashing, If unmelted material is encountered due to inadequate shear from the screw, increase the back pressure starting at 50 psi (0.3 MPa) and increasing incrementally to 300 psi (2 MPa) until satisfactory results are achieved. For best molded part performance, avoid stock temperatures in excess of 500℉ (260℃).

Adjust for part quality by use of the injection pressure and time, hold pressure and time, and finally cooling time. As a starting point, 50-75% of maximum machine capacity for both boost and hold pressure is suggested. Weigh a series of complete shots (including runner if appropriate) as the injection hold is increased to determine if the part is fully packed. When two successive shots are found to have the same weight, reduce the injection hold to the last setting. A screw cushion of 1/8 in (3 mm) is highly recommended.

Melt temperatures

The barrel and melt temperatures shown at right are good starting points. Use of preheated hand held melt pyrometers is strongly recommended to check a melt puddle in that barrel temperatures do not always reflect the true temperature of the melt. Most barrel thermocouples measure steel “barrel” temperatures instead of melt temperature.

Flow Characteristics	Melt Temperature Range, ℉ (℃)
Low flows	475-500 (246-260)
Medium flows	445-475 (228-246)
High flows	400-445 (200-228)
Extra high flows	385-420 (200-215)
Ultra high flows	370-385 (185-190)

Mold temperature

For good surface appearance of the part, mold temperatures between 70 and 120℉ (20 and 50℃) are generally used. However, the most suitable mold temperature will be determined by the individual mold, part dimension and molding cycle.

Injection fill (first stage) pressure

The fill pressure should displace 90-95% of the total shot and be the maximum pressure that can be achieved without flashing. The melt temperature employed best determines the maximum pressure to be used. Ideally, the fill pressure should initially be set at 60% of the machine maximum, 2,000 psi (14 MPa). This translates into a hydraulic pressure gauge reading of 1,200 psi (8.3 MPa) or a plastic pressure of 12,000 psi (83 MPa). The melt temperature should not be decreased to the point where excessively high injection pressure must be used.

Injection hold (second stage or pack) pressure

The injection hold pressure determines part shrinkage. It is frequently set lower than the fill pressure. The second stage injection should provide complete packing of the mold and allow the gate to freeze off.

Back pressure

Screw back pressure will produce consistent shot density and is commonly employed to improve color dispersion and melt temperature uniformity. The effect is one of increasing the amount of work going into the mix. However, screw recovery rates decrease with increased back pressure. Typical back pressure used in injection molding polypropylene is set between 50 and 300 psi (0.3 and 2 MPa).

III. Optimizing mold cycles for maximum output

The molding cycle is comprised of the following steps, each having a time component in the overall cycle time:

- Injection fill
- Mold pack
- Hold
- Part cooling
- Mold open
- Part ejection
- Mold close

Screw recovery ↓

The minimum cycle that will maximize productive output can be determined by studying each time component in the following order:

1. Injection hold time

To reduce cycle time and thereby maximize output per unit time, the injection hold time should be decreased in 1 sec. increments until a minimum time is reached (as long as necessary to freeze off the gate).

2. Injection fill time

Injection fill time should be held to a necessary minimum. Although a fast fill is often recommended for the injection molding of Pro-fax polypropylene, it has been observed in some cases that a slower fill has corrected many molded part defects.

3. Cooling time

The cooling time should be reduced in 1 second increments. The limitations to reduction of this time component are the appearance and dimensions of the molded part. The minimum cooling time can be determined by a close inspection of the parts produced after each incremental change. Part ejection temperature can be as high as 120 to 140°F.

At times, reduction of the cooling time will not be possible as the injection molding machine will be limited by screw recovery. When this occurs, the cooling time can be optimized so that the screw recovery remains the limiting component.

4. Part removal time

This time component is minimized by an incremental decrease of the mold open and close times. It can be further minimized by stripping during mold open and by an increase in ejection speed, if possible.

A careful and methodical approach to cycle time reduction following the steps just listed will help the molder minimize production costs.

IV. Use of regrind

Excess molded material (runners, sprue, gates, etc.) and rejected parts can be reground and processed in a blend with virgin material.

Regrind polypropylene should be considered, from a property standpoint, as being different from the virgin material. Depending on the severity of the molding conditions, the reground material can have a higher melt flow rate, possible discoloration (yellowness) and slightly lower physical properties such as stiffness, impact strength, tensile strength and elongation. Another issue that needs to be considered is the quality of the regrind due to potential contamination during grinding and subsequent handling.

As a starting point, it is recommended that approximately 20% regrind be used. However, the molder must experiment to arrive at the optimum blend ratio that can be molded with acceptable part performance and appearance.

Troubleshooting techniques
Common molding problems

Problem	Possible causes
■ **Short shot (parts not filling)**	- Insufficient injection fill speed - Insufficient pack pressure - Insufficient injection time - Insufficient shot size - Unbalanced multiple cavity mold - Foreign material clogging nozzle and/or gates - Stock temperature too low - Runners, gates, or vents too small - Mold temperature too low - Undersized barrel heating capacity - Excessive wear of screw, barrel or check valve
■ **Mold flash**	- Excessively high injection pressures (fill or pack) - Material on mold surface - Low clamping pressure - Stock temperature too high - Resin melt viscosity too low - Shut-off faces of mold mismatched
■ **Excessive shrinkage**	- Cure time too short - Pack pressure too low - Mold or stock temperature too high - Insufficient injection pressure - Runners or gates too small - Poor part design, varying wall thickness
■ **Warpage**	- Part ejected too hot - Improperly balanced core and cavity temperatures - Inadequate or poor location of knockout mechanism - Overpacking in gate area because of high injection pressure - Molded-in stresses due to low stock temperature or mold too cold - Improper part design, nonuniform walls - Improperly balanced multiple gates - Flow too long, insufficient gates
■ **Sink marks**	- Insufficient injection pressure - Insufficient dwell or hold time - Melt or mold temperature too high - Poor part design, non uniform walls and/or excessive wall thickness - Improper gate location or design

Troubleshooting techniques
Common molding problems

Problem	Possible causes
■ **Brittleness**	- Degraded material from barrel (excessive melt temperature) - Contamination by other polymers - Improper design; inadequate radii at corner, notch, or thread - Use of improper color concentrates (non-compatible carrier resin) - Voids - Stock and mold temperatures too low
■ **Flow marks** **Weldlines** **Low gloss** **Rough surface**	- Stock temperature too low - Mold temperature nonuniform or too low - Mold fill too fast or too slow - Excess mold lubricant - Scratched or dirty mold surface - Fill speed and/or packing time too low - Poor pigment dispersion - Inadequate venting - Improper gate location or design
■ **Erratic quality**	- No cushion or back pressure - Unbalanced multiple cavity layout and runner system - Non-uniform feed temperature - Non-uniform cycle - Unbalanced multiple cavity layout and runner system - Undersized barrel; insufficient volumetric and heating capacity
■ **Voids**	- Failure to fill mold completely (see Short Shot) - Poor venting of mold, particularly around projections - Improper location of gate - Fill rate too rapid (trapped air produces short shots) - Excessive part thickness (greater than 1/4" or 6.3 mm)
■ **Sticking in the mold**	- Overpacking; injection pressure too high - Underpacking => excessive shrinkage - Insufficient cooling - Highly polished core surface => draw polish - Insufficient knockout action - Surface irregularities in the mold - Insufficient core and wall tapers - Undercuts