INTRODUCTION

The intent of this technical brochure is to provide molders with a general insight into the characteristics of polypropylene and aspects of design for optimum results. Prospective molders should consult with a CPC technical representative for questions which can't be answered with this free guide.

DESIGNING PARTS FOR POLYPROPYLENE

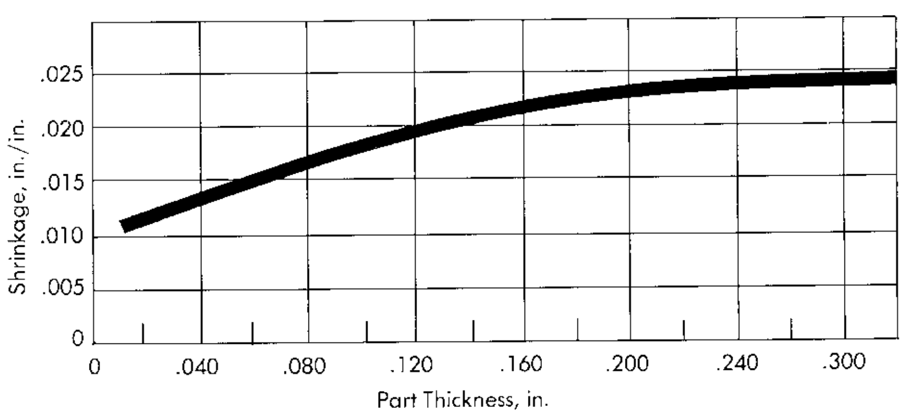
Part Design Considerations

Wall Thickness — Effect on Shrinkage

The wall thickness has a direct effect on polypropylene's mold shrinkage. This contraction of the injection molded part usually continues for approximately 24 to 48 hours or longer after molding.

As shown in Figure 5, heavy wall sections experience greater shrinkage. This provides the fundamental argument in favor of designing molded ports with constant wall thicknesses, since a part with varying wall thickness will have varying shrinkage, and this, in turn, can lead to stressed and potentially warped parts.

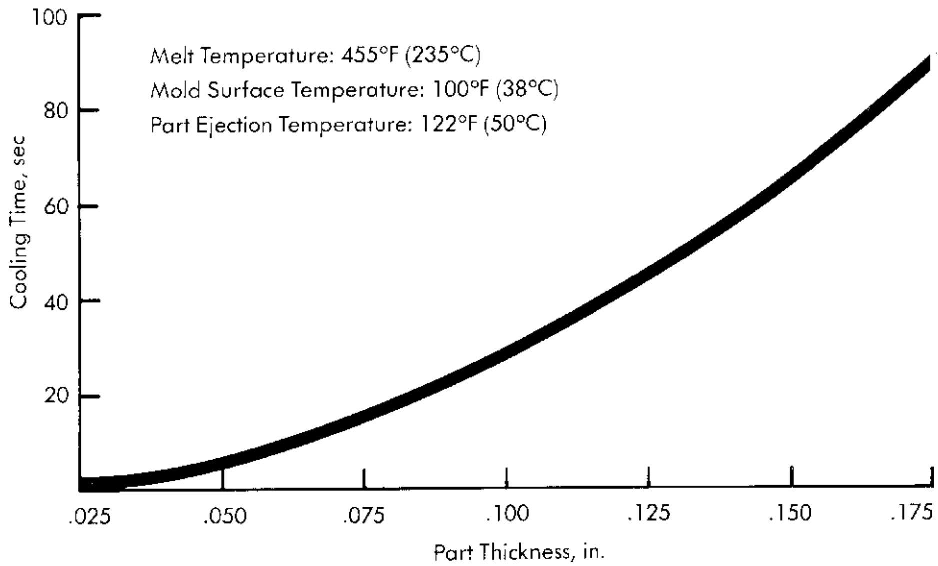
Figure 5
Pro-fax Polypropylene Shrinkage Variation With Thickness



Note: Shrinkage values given here are the maximum to be expected under most conditions. 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.

As shown in Figure 6, 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.

Figure 6
Cooling Time vs Part Thickness



Nucleation

Polypropylene is nucleated to produce clarified, dimensionally stable ports. Most of the postmolding shrinkage occurs in nucleated parts in the mold. The mold, therefore, becomes a cooling jig stabilizing the port to the mold shape, thus preventing worpage. However, the ports can be difficult to strip owing to shrinkage on cores. Nucleated ports olso tend to freeze in molding stresses and may be less tough.

Flow Considerations

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-high flows, while thick-walled parts with short flow paths allow the use of medium to low flows.

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 strength of the hinge.

Guidelines

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

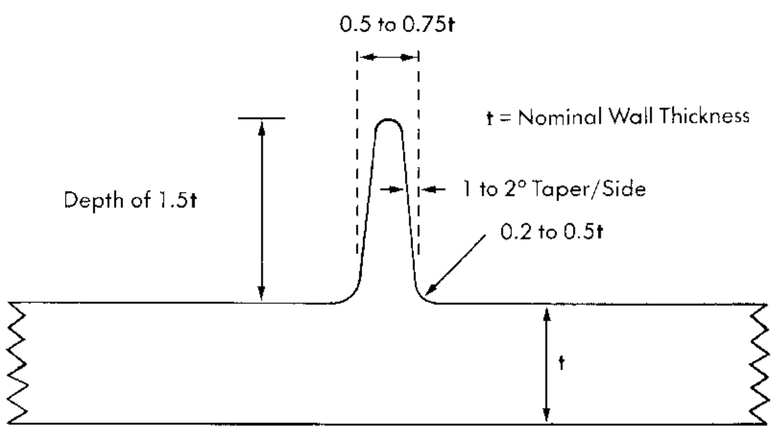
Ribs

Ribs are intended to stiffen and strengthen the molded part while maintaining a minimum wall thickness. Furthermore, they help cantral 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 20 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 0.5 to 0.75 (moximum) times the adjacent nominal wall thickness. The rib should further be tapered ito 2° per side, and should have a depth of 1.5 times the nominal wall. Deeper ribs can be used; however, they usually require a thicker base to allow for the draft angle. Ejection problems may increase when deeper ribs are used.

Figure 7
Recommended Rib Design



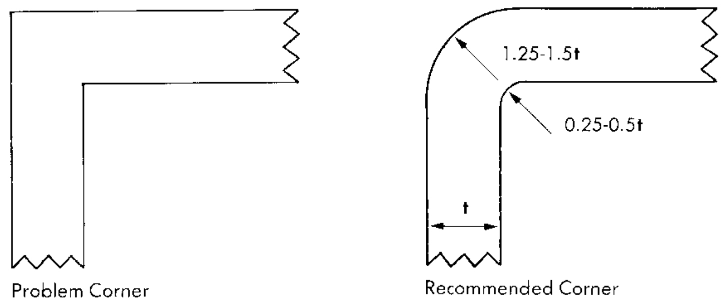
As can be seen in Figure 7, a properly designed rib is thin and shallow rather than thick and deep.

Radii

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

The inside corner radii should be 0.25 to 0.5 times the wall thickness. The outside corner radii should be 1.25 to 1.5 times the nominal wall thickness. Applying these guidelines helps to maintain a constant wall thickness, as shown on page 11.

Figure 8
Correct Wall Thickness at Corners



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 1° per side is adequate, although more is recommended for easier port release. Textured surfaces require an additional draft of 1° per side for each 0.001 in. (0.025 mm) of texture depth.

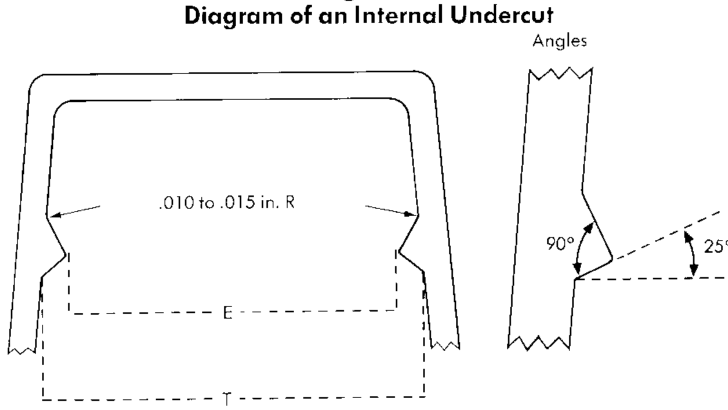
Undercuts

Undercuts should be avoided. However, for articles (such as closures) that require threaded undercuts, the undercuts should be designed with a lead-in angle of approximately 25° to facilitate the part's strippability. 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 (see Figure 9) is determined as the percentage difference between the outer diameter, '97' and the inner diameter, "E."

Undercuts, which cause more than 5% deformation, can permanently damage the part. The percentage of undercut size represents the allowable percentage rate of deformation (strain) of the undercut.

Figure 9
Diagram of an Internal Undercut



$$\text{Undercut Size (\%) or Strain (Deformation) Rate (\%)} = \frac{T-E}{T} \times 100$$

The best location for concentration of mold cooling lines is in the area of the hinge. At this point, there will be additional frictional heat, and the mold design requires a relatively thin extension of the mold steel in this *area.* In some parts, it may be necessary to consider other means of cooling in the hinge area, because overheating can result in lamination and hinge weakness.
In o port with a living hinge, the location of the gate or gates is as important as proper cooling in the prevention of lamination or weld lines in the hinge. The next section will help molders design the best possible part.

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” can be prevented or solved using the proper gate and location.

A number of gate types are located at the parting line. The usual types are: (1) flat gate — a simple rectangular channel; (2) full round gate — similar to a channel gate, except round and cut into both mold halves, which can be tapered; (3) fan gate — where the flow diverges to a wide, shallow intersection with the part; (4) flash gate — o thin membrane formed between a runner and a long side of the port; (5) ring gate — a modification of a flash gate that would surround a tubular port; and (6) diaphragm gate — o flash gate that feeds the interior of a tubular part. In general, parting line gates require mechanical trimming.

Gate designs that do not intersect the part at the porting 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 post tapering away from the port, which must be snipped away. If appearance is important, it can be machined away. Pin gates are used with both three-plate molds and hot runner systems. They normally avoid trimming or degating operations. The subgate is fed by a porting 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.

For parts weighing a few ounces, a gate area of .001 in.2 will be adequate, while for large parts weighing 10 lbs or more, several gates that total 0.5 in.2 may be required. For similar performance, gates must have identical land length (the parallel wall section as the gate meets the cavity) as well as area. Land lengths for polypropylene should be 0.020 to 0.030 in. Gates without lands 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 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

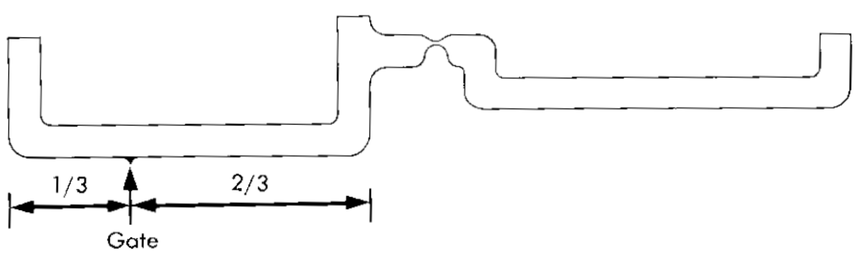
Because the gate area is often highly stressed, it should be located so that the product's properties and appearance are not adversely affected.

Gate location should:

- Ensure a balanced flow (rapid and uniform filling) in the cavity so that certain areas of the part are not overpacked
- Ensure mold fill under realistic temperatures and pressures
- Minimize weldlines as much as possible, or position them in noncritical areas
- Prevent "jetting" by positioning the gate so that material flow is smooth and uniform
- Avoid air entrapment
- Gate in the thickest section and direct mciterícil flow from thick to thin sections

For parts with an integral hinge, the gate should be located beyond the centerline of the cavity, away from the hinge. This is particularly true for the molding of shallow boxes, where the most suitable gate should be located as shown below:

Figure 11
Correct Gate Location for Shallow Boxes



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 burn marks, 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 weld lines

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. (00254 to 0.0508 mm); 0.0015 in. (0.038 mm) is typical.
- Land Length: 0.040 in. (1.016 mm).
- Width: As required.

Cooling Requirements

To achieve 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 in. (11.1 to 14.3 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

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

1. Increase the coolant flow rate.
2. Reduce the coolant temperature.
3. Clean out the buildup of sediment in the channel.

As a recommendation for chiller capacity, allow 1 ton of capacity for every 35 lbs of polypropylene to be processed per hour.

Multicavity Mold Design

If a multicavity mold will be used vs. a single-cavity mold (based on machine size, part quantity, delivery period, and molding cost price), 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 reduces runner volume by varying the size and shape of the runners and hot runners, but if possible, do not balance by varying the gate sizes.

Ultimately, the ideal balanced mold should deliver polypropylene to each gate under identical conditions (i.e., temperature and pressure).

Unbalanced molds usually cause overpacking, warpage, differential shrinkage, brittleness, dimensional inconsistencies, and parts that stick to the mold.