Design Tips
for Rapid Injection Molding
Volume 8
Design Tips categorized by topic

<table>
<thead>
<tr>
<th>Page</th>
<th>TABLE OF CONTENTS</th>
<th>Material selection</th>
<th>Design guidelines</th>
<th>Quality assurance</th>
<th>Understand the process</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>Breaking down gates</td>
<td>●</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>The skinny on living hinges</td>
<td></td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>7</td>
<td>Draft on tap for stress</td>
<td></td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>9</td>
<td>Designing outside the box</td>
<td></td>
<td></td>
<td>●</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>What Color is your Prototype?</td>
<td></td>
<td></td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>13</td>
<td>Choosing sides</td>
<td></td>
<td></td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>15</td>
<td>The Protomold guide to looking good</td>
<td></td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>17</td>
<td>Shining some light on lenses</td>
<td></td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>19</td>
<td>Through thick and thin</td>
<td></td>
<td></td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>21</td>
<td>Round like a wheel, not like an egg</td>
<td></td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>23</td>
<td>Mold finish vs. part finish</td>
<td></td>
<td>●</td>
<td>●</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>Being transparent</td>
<td></td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
</tbody>
</table>

©2012 Proto Labs, Inc. All rights reserved.
Let’s face it; a gate is a necessary evil, a break in the otherwise continuous surface of an injection mold. While its vestiges are typically removed after molding, either automatically or manually, it leaves its “footprint” on the finished part. Without it there would be no part, so since we can’t do away with gates and because they can impact both the moldability and quality of the part, there is good reason to consider their type and location in designing parts. Of course, even if you don’t think about gates when designing your part, you will have a chance to approve their type and location when you confirm your Protomold order.

At Protomold, we use tab, hot tip, or pin gates depending on the shape and size of the part and the resin being injected.

**Tab Gates**
The tab gate is the most commonly used of the three types. It consists of a trapezoidal block milled into the parting line on an exterior surface of the part (see Figure 1), and is usually placed in a thicker wall section rather than thinner areas. Its position usually makes it easy to trim off, and while it typically leaves the largest vestige of the three gate types, its position at the parting line affects only the edge of the part, where it usually does not interfere with function or cosmetics.

The tab gate is easy to manufacture, maintain, and process, and there is typically a great deal of latitude in its placement along the parting line. Its design helps confine stress generated during ejection to the tab, which is trimmed off after molding. Also, being relatively large, it is ideal for use with filled resins, which can be difficult to inject through smaller gates. For all these reasons the tab gate is the first choice among gates.

**Hot Tip Gates**
The hot tip gate is positioned near the center of a part rather than at the edge. This reduces the distance resin must travel to fill the mold and helps center the clamping force the press applies to the mold. Figure 2 shows an image from a sample Protomold order confirmation with a hot tip gate location. The yellow cylinder is a cavity milled into the A-side mold half to which a single-drop hot tip will be bolted. Hot tip gates are always on the A-side.

The hot tip gate is often used on round or domed parts to achieve radial flow rather than the linear flow of a tab gate. Also, because this gate takes up no space at the edge of the mold cavity, it can be used on parts approaching Protomold’s maximum part footprint size.
The gate leaves a small raised bump, typically .060 to .080 inches in diameter and .010 to .020 inches high, that can be trimmed flush to .005 inches proud. Depending on part geometry and resin type, a hot tip gate can also leave “blush” or flow marks in a circular pattern around the gate site. These can be problematic because they are typically on the cosmetic surface of the part. Another problem is the potential for degrading resin in small shot sizes. And finally, because it is a small gate, a hot tip can become plugged if used to inject resins with high glass fill content.

Pin Gates
The pin gate is the least common of the three gates. It is typically used for complex parts that cannot have gate vestiges on the edges where tab gates would be or on the cosmetic side of the part where hot tip gates are located. It is the only one of the three gate-types that is located on the B-side of the mold, typically the non-cosmetic surface.

Instead of introducing resin into the mold cavity where the part is formed, a pin gate injects resin into the ejector pin channel (see Figure 3). The ejector pin, shown in yellow below, is shortened, and resin is injected from the cone-shaped “tunnel” gate through the empty section of the ejector pin channel, shown in purple, and into the mold to form the part. Once the resin has hardened, the ejector pin pushes against the cylindrical post in the pin channel to eject the part from the B-side mold half. The ejector pin shears the post from the tunnel gate, leaving the post rising from the surface of the ejected part. The post is later removed in a manual process. Because it injects resin through an ejector pin hole, this gate creates a vestige on the finished part like an ejector pin but trimmed flush to .005 inches proud instead of the standard ejector pin that is flush to .005 inches recessed into the part.

The use of the pin gate is very geometry-dependent, and while it can produce excellent results, it can be challenging to process. The cone-shaped tunnel gate precludes the use of many glass-filled resins, and the shearing of the cone from the post during ejection can place significant stress on the mold. Finally, the angle and depth of the cone rules out its use with many geometries, such as housings or other cored-out parts.

Clearly, gating can involve complex choices, but Protomold will select the gate type and position that seems best suited to your design and resin. You, in turn, will have the opportunity to review the gate placement on your order confirmation. If you have questions or concerns, you can contact our Customer Service Engineers at 877.479.3680 to discuss options.

Figure 3: This gear is an excellent example of a part that used a pin gate. The final part, on the right, shows the post from the tunnel gate, which is removed in a manual process.
The skinny on living hinges

A living hinge is a thin band that allows a plastic part to be bent without breaking. Applications include parts that bend once for assembly—a flat part that folds into a cube for example (see Figure 1)—or a cover that may open and close hundreds, even thousands of times (see Figure 2).

We’ve often pointed out the consequences of forcing melted resin through a thin area of a part into a thicker area. The most serious of those consequences is the possibility of voids in the thick area beyond the “bottleneck.” To create a living hinge, you must do exactly what we’ve warned you against: force resin through a very thin area into a thicker section of the part (see Figure 3).

The only alternative would be to have resin enter through two gates and meet at the hinge. Unfortunately, this would create a knit line (a weak area in any part but especially problematic at a thin area) precisely where you want the part to be strong enough to bend without breaking. So, since living hinges are so very useful, the question is how to avoid—or at least anticipate—problems if we choose to include them in a design.

■ Rule One: Scrupulously follow the guidelines for hinge design. These can be found in the earlier design tip, Night of the living hinge—see volume 4, page 13.

■ Rule Two: Follow all the other guidelines for molded part design; just because you are bending one rule—no pun intended—doesn’t mean you can push your luck on others.

■ Rule Three: Choose an appropriate resin. Choose one that flows well through thin areas and that, assuming a properly designed hinge, can handle bending without weakening or breaking. The ideal resin for living hinges is polypropylene; second choice is polyethylene. Depending on your application, there may be other resins with limited applicability, but if you absolutely require the strength of a glass-filled nylon or the durability of acetal, for example, you probably shouldn’t plan on using a living hinge.

Continued on next page...
**Rule Four:** Prepare for possible cosmetic consequences. A living hinge is like a construction bottleneck on a freeway; during injection it's going to increase pressure on the upstream side of the hinge and decrease pressure on the downstream side. Also, because of the mold's increased surface-to-volume ratio at the hinge, resin is going to cool significantly at that point. Consequences downstream of the hinge may include sink, flow marks, and bubbles. (Bubbles may not be apparent in an opaque resin, though they may affect strength. If you use a clear resin, bubbles will be visible.) Consequences upstream of the hinge may include flash caused by the increased pressure needed to force resin through the thin area.

Got questions about hinges, living or otherwise? Proto Labs Customer Service Engineers are standing by to help at 877.479.3680.
Draft on tap for stress release

Long before any of us designed, produced, or even thought about plastic parts, we experienced draft in the form of a Popsicle® or similar frozen treat. An ice pop’s clean, icy surface is achieved the same way we create the unmarred surfaces of plastic part, by drafting—tapering—the sides of the mold so that the surface of the molded object pulls away from the mold walls during ejection (see Figure 1). If the mold sides were straight, removal from the mold would be difficult and the surface of the pop could be marred as the ice was pushed out of the mold. It would still taste the same, but would suffer cosmetically.

The molding of ice pops is complicated by the fact that water expands as it freezes, a problem that is overcome by leaving the handle end of the mold open so the expanding ice has somewhere to go. If the mold were closed, the expansion of water as it turns to ice could result in what might be called “ice flash,” much like what occurs when a plastic injection mold is overpacked.

Unlike pops, many plastic parts include features formed by cores protruding from the B-side mold half (see Figure 2). While shrinkage of the cooling resin could, in theory, cause the outside surface of the part to pull slightly away from the A-side mold half; that same shrinkage can cause the part to tightly grip the core that formed the feature. This is best addressed through the use of a drafted part (resulting in a drafted mold wall), which effectively causes the part to move away from the mold wall as it is ejected.

Molders will work hard to prevent shrinkage of cooling resin away from the mold half that forms a part’s outside surfaces, as it may result in out-of-tolerance dimensions. They do this by continuing to inject resin into the mold as it cools, forcing the solidifying, cooling resin farther into the mold. In other words, a mold that is “full” with heated resin may be only 95 percent full once that resin starts to cool. Left at 95 percent of capacity, the resulting part might be successfully ejected without standard draft. It would, however, run the risk of surface sink, voids, and failure to pick up proper texture from the mold walls. The addition of the final five percent of the mold’s full capacity reduces these risks at the same time that it reduces shrinkage away from the mold wall. Proper draft prevents this tight fit from hindering ejection while letting molders avoid the cosmetic problems that come from less-than-optimal filling.

There are two other reasons for incorporating draft into a design. The first is to prevent damage to the mold wherever metal slides against metal, as in a sliding shutoff. The second is to allow end mills to make deep, narrow cuts to create tall ribs. (Check out our sliding shutoff video on YouTube.)
The reason for drafting sliding shutoffs is simple. Without draft, the metal faces would quickly wear, damaging the mold and allowing flash to form in the spaces between worn mold surfaces. Drafting the metal faces minimizes wear as the mold opens and closes.

Deep, narrow cuts require the use of long end mills, and the farther the cutting tip is from the chuck of the mill, the easier it is for the cutter to be pushed out of position as it spins. This can cause chatter, resulting in gouging of the piece being milled, and it can actually break the end mill. A wider rib allows the use of thicker end mills, which can withstand the side-load and maintain stability as they cut. If the rib must be thin, however, drafting its sides allows the use of a tapered end mill, which will be more stable than a straight one with the same size cutting tip. (Note: If your design truly needs tall, thin ribs with minimal draft, our process also supports the selective use of EDM to make that possible.)

Don’t forget to draft what sometimes seems inconsequential. Text and other similar geometry like logos and very shallow features have an amazing ability to stick to a mold and cause pulling. This forms small pieces of standing material that are sharp and can affect the appearance of text and cosmetic details. A little draft goes a long way in releasing resin from the finer detail of the mold.

If you submit parts with insufficient draft, the design analysis in the ProtoQuote® interactive quote will point out areas where greater draft is required. But while you can add draft late in the design process, your design will benefit if you consider the need for draft right from the start. General guidelines for draft are:

- at least 0.5 degrees on all vertical faces
- 2 degrees to provide a margin of safety in most situations
- 3 degrees minimum for a shutoff (metal sliding on metal)
- 3 degrees required for surfaces with light texture (PM-T1)
- 5 or more degrees required for heavy texture (PM-T2)

Product design presents you with a series of choices involving the function and aesthetic of the product, the methods of manufacture to be used to realize the product, and physical, financial, timeline, and other constraints on the choices. This month’s tip describes some of the choices facing one designer during a product’s development. We hope you will gain some insights you can use in your own development projects.

The product is Reptangles™, a building toy consisting of identical, faceted turtle-shaped blocks that can be connected in almost limitless ways to create complex three-dimensional designs. The concept has been patented as “Multifaceted Nesting Modules.”

The project had three major design challenges: developing the shape of the plastic turtles; creating the connectors—56 per block—that allow the turtles to be connected in a variety of configurations; and designing the blocks so they could be produced in simple and inexpensive straight pull molds in order to keep the product cost low.

The connectors had to be located in a variety of orientations on a faceted block (see Figure 1). This was part of the reason that a simple friction fit, like that of LEGO® blocks, was not workable. A friction fit requires extremely tight tolerances that can be lost with wear, and while LEGO blocks are typically stacked vertically so that gravity helps keep them connected, Reptangles can be connected in virtually any orientation, and thus connections can be pulled in almost any direction. Designer Jonathan Stapleton quickly realized that his connectors would have to snap together to resist separation, but still be separable. He also saw that, in making connections, the faces would not necessarily approach one another in a direction perpendicular to the faces, but could move obliquely toward one another as the faces met.

Moldability was quite a challenge. Because the turtles were to be three-dimensional and hollow, they would be molded in two parts—top and bottom, as in Figure 2—and then joined to form the finished block, but each half still had to be produced in a two-part mold. With faces positioned at 45°, 90°, or 135° to the direction of mold opening, connectors would have to be specifically designed to avoid undercuts.

Through CAD modeling and prototyping, Stapleton found a solution based on triangular connectors. Right-angle arches on one face would fit into slots on the mating face, and catches in the walls of the slot would create an interference fit (Figure 3). The triangular shape of the male connector allows insertion in any direction within a 90-degree arc.

Continued on next page...
This design requires a resin with enough flex to allow the catches in the slots to move slightly during attachment and separation of the mating parts but still hold tightly enough to prevent accidental disconnection.

Having solved the connector problem, Stapleton moved on to issues of molding. Dividing the finished block into top and bottom halves allowed connectors to be formed by the interaction of the A- and B-side mold halves. The underside of the male connector is formed by a protrusion through the part wall of the B-side mold half, which meets the A-side mold face in a sliding shutoff. As seen in Figure 4, this allows the molding of male connectors on faces both perpendicular and oblique to the direction of mold opening. Similarly, the female connectors are formed from the “in-side” of the part by protrusions of the B-side shutting off against the surface of the A-side mold half.

The designer recognized that, if the female connectors were designed to follow the contours of the male connectors, those on the oblique faces would represent undercuts. He solved the problem by aligning the potentially problematic surfaces of those connectors with the mold opening direction (see Figure 4). Because the interference fit is accomplished by the catches of the female connector grabbing the loop of the male connector, this did not impair connector function.

Finalizing the part design was a multi-step process. After CAD modeling and prototyping in wood, Stapleton recognized that the only prototyping methods that would positively confirm moldability were injection molding. The first set of Protomold’s injection molded prototypes showed the need for minor modifications in the design. Fortunately, the needed changes involved increasing the size of some features (“adding plastic”), which could be achieved by modifying the first mold rather than making a new one. The final product was licensed to a toy company and is now on the market, and the designer has filed a patent application for the connectors themselves.

For a detailed case study of this product, go to the geometoy case study at www.protomold.com.
What color is your prototype?

Injection molded parts, for all their many shapes and sizes, all start out pretty much the same—as small plastic pellets. The base colors of those pellets vary somewhat depending on the resin, but they all fall somewhere on a monochromatic scale from clear through various shades of natural to black (see Figure 1). Turning those dull little pellets into the rainbow of colors that comes out of molding presses requires the addition of colorant, which can be a fairly simple or fairly involved process. The complexity depends on how picky you are about your color.

If you want to see all the choices we offer, go to bit.ly/PMcolors. Of course, how closely the final color resembles the color you see on your screen may depend on the screen itself, but it will certainly be red. If, on the other hand, you want a very specific color, say the blue of your corporate logo or the color-matched cap of a spray-paint can, the process becomes more complicated.

There are basically two ways to get colored injection molded parts. The simplest is to mix dye pellets into a batch of base resin pellets (see Figure 3). As the pellets are heated and compressed for injection into the mold, the base resin and dye pellets melt and mix together before being injected into the mold.

The other way is to order pre-compounded pellets in the exact color you want; for example, a standard PMS color or match to a sample swatch (see Figure 4).

Continued on next page...
There are several things you should keep in mind if you choose to have Protomold add the colorant.

1. As stated above, Protomold does not match specific colors.

2. The effect of base resin shade on final color is greater on light colors. In a pre-compounded color, this can be adjusted for in the compounding process, but may be noticeable when you use Protomold’s stock colors.

3. Because dye pellets are mixed with base resin pellets in a “salt and pepper” mix, there is a chance of “swirling” in the resulting parts (see Figure 5). The degree of swirling depends on a number of factors. “Hot” colorants—red, orange, yellow—tend to exhibit a higher swirling risk than cool colors like blue and green. Also, large parts, because they use more of the contents of the chamber in which resin pellets are melted and mixed, are less likely to show swirling than small parts.

4. Unless you use pre-compounded colored resins, there is a risk of inconsistency from lot to lot and from part to part. This is one of the reasons that we recommend our standard ratio of three percent (3%) colorant (see Figure 6).

5. Certain base resins “don’t play well with others” when it comes to colorants. For these—high-temperature nylon and polycarbonate, for example—Protomold stocks a limited variety of resin-specific colorants. These allow you to achieve color but further limit your choice of hue.

6. Finally, certain resin characteristics—UV protection, flame retardant, and medical or food compatibility—can be affected by the choice of colorant. Protomold does not stock colorants specifically made to maintain these characteristics.

All of these problems have a solution: the use of pre-compounded resins purchased from specialized vendors. Pre-compounding consists of mixing colorant with base resin, melting and extruding the resin, re-pelletizing the resulting mix, and then repeating the process until a thorough mix is achieved. If necessary, the mix can be adjusted during the process to match the customer’s exact specifications. Vendors like RTP Plastics, PolyOne, Accek Color, Chase Plastic, and Compounding Solutions can match virtually any color in a wide variety of resins, and can preserve critical resin characteristics in the process (see full contact list below). Because the color is evenly distributed among the pellets, there are no issues of swirling or part-to-part inconsistency. Protomold can use these pre-compounded resins to produce parts, delivering all the benefits of rapid injection molding along with the exact color and characteristics you need.

In short, if you need approximate colors, for general appearance or any other reason, Protomold can provide a range of colorants at no charge. For specific colors, maximum consistency, and protection of specialized resin characteristics, outside vendors can provide the colored resins and Protomold can deliver the parts. If you have any questions, feel free to contact our Customer Service Engineers at 877.479.3680.
Choosing sides

When you start looking into injection molding, one of the first things you notice is that everyone talks about the “A-side” and the “B-side.” These terms refer to one side or the other of an injection mold. When a mold is manufactured to make your part, some of your part’s outer surface is created by the A-side and some by the B-side. A-side and B-side are such fundamental concepts for injection molding that nobody bothers to explain what these two terms really mean. There is a complex web of factors and consequences around which side of your part is assigned to the A-side, and which to the B-side. Understanding the implications can help you design better parts and to avoid unpleasant surprises down the road.

Most of the constraints and attributes around “sidedness” in injection molding track back two basic factors. First is physics: plastic shrinks as it cools. Second is the conventional design of injection molding machines. Almost all injection molding presses are built so they inject molten plastic into one side of a mold (conventionally called the A-side), and have the part ejection system in the other side of the mold (conventionally called the B-side). In Figure 1, the injection unit of the press is on the left side, and the clamping unit (which incorporates the ejection system) is on the right side.

One of the main drivers for choosing which side of your part is the A-side and which is the B-side is ejection. It seems plausible that when the mold opens after forming your part, it will just tumble out to make room for the next cycle. Such is not the case, however. Since the plastic shrinks as it cools, it shrinks around any convex parts of the mold, and it holds on tight. And, except for a few rare geometries, your part will hang on to both sides of the mold. Injection molding presses are designed with this in mind, though, and the press can use quite a bit of force to pull the mold open. Your part will generally stay in the half of the mold that has the most convex surface area.

Continued on next page...
If that side is the B-side, no problem, the ejector system will push the part out of the B-side to get ready for the next cycle. If that side is the A-side, then everything stops until the molding process technician can figure out how to pry the part out of the A-side without damaging the mold.

Part surfaces can be extremely convoluted, with ribs, bosses, cores, through-holes and other features adding to each side’s tendency to grip the mold. In some cases, identifying the A- and B-sides requires both software tools and a fair amount of experience and intuition on the part of Protomold design staff. Even so, sometimes minor changes to the mold design or manufacturing process are required to ensure the part will stick to the B-side.

Whether you design your part to have a particular A/B orientation or simply approve the orientation chosen by Protomold, the orientation of your part to the mold halves will make a difference in cosmetics.

Depending on gate type, the A-side of a part may show vestiges of the gate, particularly if a hot tip gate is used. This is significant because the A-side is often the cosmetic side of a part, e.g., the outside of a case or shell. These vestiges can be anticipated and camouflaged or covered, for example, with decals.

The B-side will typically show ejector marks. These are usually less critical, as the B-side is often the hidden, non-cosmetic side of a part. There are exceptions—a concave plastic tray designed to be set into a surface, for example—in which case, the marks can be anticipated and treated in the same way as gate vestiges.

Your free ProtoQuote® interactive quote will provide a design analysis that will show how Protomold has decided to assign sides. In your quote, surfaces formed by the mold’s A-side are colored green, those formed by the B-side are blue, and, if present, surfaces formed by side-actions will be shades of pink. You will be asked to sign off on these aspects of the design, along with the gate and ejector placement, as part of the order confirmation process.

There is sometimes latitude in part orientation, and simple changes to your part design will often allow different orientations. As always, questions about side assignment or any other aspect of part design can be answered by our Customer Service Engineers at 877.479.3680.

To learn more about specific mold and part features referred to in ProtoQuote interactive quotes, and the operation of the mold itself, sign up to receive a no-cost Protomold Demo Mold sample at www.protomold.com/demomold.
First, the bad news: injection molded parts are subject to a host of cosmetic flaws including sink, gate vestiges, ejector pin blemishes, drag marks, texture flaws, knit lines, burns, flash, and inconsistent coloring. The good news is that adherence to basic design rules will eliminate many of these problems. Thoughtful planning will further reduce the risk. And judicious prototyping will allow you to eliminate virtually any avoidable problems that remain.

The first step is to determine just how important cosmetics will be to your design. In a faceplate, looks may be your first consideration. In an internal part, they may be your last, freeing you to focus on other matters like function and cost. In some cases, improving appearance will require tradeoffs, but this is not always the case. Good function and good manufacturability often go hand-in-hand with good appearance.

For example, proper attention to draft doesn’t just ensure ease of ejection; it prevents unsightly drag marks (see Figure 1). Maintaining even wall thickness helps prevent functional weakness due to incomplete filling and poor fit due to warp, but it also helps prevent unsightly sink in thick areas and the flash and burn that can result from overly high injection pressures needed to force resin though thin sections. In short, following standard design guidelines will help prevent many cosmetic flaws.

This is where the tradeoffs begin, where gains in one area may require sacrifices in another. Start by ordering your priorities. Certain requirements may impact appearance by limiting your choice of resins. Glass filled nylon, for example, is very strong but cannot be given a smooth finish regardless of how much the mold surface is polished. Similarly, a TPE like Santoprene™ provides excellent elasticity, but will have, at best, a matte finish. Acetals like Delrin™ offer excellent resistance to solvents and wear due to friction, but can develop an “orange peel” finish as they cool. In short, if your requirements are very specific and limit your choice of resins, they may also limit your cosmetic options, at least as far as finish is concerned. If, on the other hand, your material choice is less constrained, resins like ABS and polycarbonate are easy to mold and will give you precisely the finish designed into your mold.

Continued on next page…
Many cosmetic decisions cross the boundaries of material and design. Knit lines form where cooling flows of resin meet, typically after parting to flow around a core in a mold or where flows from multiple gates meet (see Figure 2). Choice of resin can impact the likelihood of knit lines, but even if they do not weaken the part, they can leave an unsightly line. How visible it is may depend on the color of the resin. Light reflection can make knit lines in black resin very apparent, while the exact same line in white resin will be far less obvious.

There are other addressable issues that affect specific types of resin. Delrin, as mentioned above, can “crinkle” as it cools, a problem that can be minimized by avoiding excessive or inconsistent wall thickness. As polycarbonate cools it can shrink and leave internal voids where thick walls meet. In an otherwise well-designed part these may not cause functional problems, but they can be a cosmetic problem in clear resin in which the “bubbles” are visible. If you must use a clear polycarbonate, the mold can be redesigned to shorten the distance resin must travel, allowing the press to “pack” the mold before the resin cools too much to fill the void.

Resins that are good at filling small details in a mold can also have an increased tendency to “flash,” i.e., to force their way into the parting line of the mold, leaving material that must be trimmed in a secondary operation. A mold in which material must be forced through thin areas will require higher injection pressure to fill the mold, which, in turn, will increase the likelihood of flash. The problem can be addressed either by eliminating the mold features that require higher injection pressure or choosing a resin that is less prone to flash.

Finally, there are issues of mold design—gate and ejector placement—that impact a part’s final appearance. Protomold’s designers will endeavor to minimize the kinds of problems described above, but all molds require gates and ejectors, and these will leave their marks on the finished parts. If this could be a problem, you should bring it to the attention of our engineers before your order is finalized.

Clearly, there are many issues that can impact cosmetic outcomes. Some may be identified by the design analysis included with your ProtoQuote®. Others involve “cross-coupled” variables and are discussed in more detail in a free cosmetics white paper available on our website. Still others may be difficult to anticipate and will only show up when prototypes are made, allowing you to try other resins or, if necessary, redesign your part before committing to full-scale production.

As always, you can discuss questions with Protomold Customer Service Engineers at 877.479.3680.
Shining some light on lenses

Previously we talked about visual characteristics of parts from a cosmetic, as opposed to functional, standpoint. Now we will address some visual characteristics that actually are the function. We’re referring to the optical properties of parts that are made of clear resins like polycarbonate and used as light covers, lenses, and lightpipes. The ability of these parts to perform optically depends on a combination of material, shape, and internal clarity. To determine whether they can be made using the Protomold process, you’ll need to understand our capabilities and limitations.

Our rapid injection molding process is specifically designed to produce very good injection molded parts, quickly and cost-effectively. It is uniquely suited to most prototyping applications and many production applications as well. In some optical applications, however, performance can be impaired by small flaws that would not ordinarily be a problem. Small imperfections like internal bubbles, minor surface irregularities, flow lines in thin sections, suspended particles, gate vestiges, and slight knit lines would have negligible impact on the function and even the cosmetics of most parts. But since they can affect the passage of light through clear resins, they can seriously impact the functionality of optical components.

Some of these imperfections may be preventable through careful design, but others may be unavoidable. For example, a widely-used clear resin (like a polycarbonate) shrinks significantly as it cools. This increases the likelihood of internal bubbles and can disrupt the passage of light through a part. Similarly, while uniformity in part thickness helps moldability, lenses often require variations in thickness and these can create flow problems or lead to sink as the part cools. And because Protomold does not offer precision grinding on mold finishes, part surfaces may be less than optically perfect.

While the Protomold process includes several steps designed to identify and head off problems, they are not designed to identify optical issues. In fact, Protomold has no way of knowing that the part being analyzed will have an optical function, so while the design analysis included in ProtoQuote® can identify molding problems, it may accept parts with optical flaws as perfectly moldable. ProtoFlow®, another Protomold analytical tool, considers choice of resin in detecting potential mold-fill problems; however, like ProtoQuote, it cannot anticipate potential optical flaws.

High quality optical production requires specialized facilities and handling. At Proto Labs, parts are produced in an open manufacturing area where there is potential for minor particle contamination that could impact optical performance. Also, we do not offer special packaging to protect optical-quality surfaces in transit. Nonetheless, we do successfully produce optical parts for clients who understand the limits of our process.

Despite its limitations, rapid injection molding can be useful in prototyping optical components even if the prototypes may not have all the optical properties of production parts. Optical properties are generally fairly predictable and may not need prototyping, but those parts may still need to be tested for fit and structural integrity. And, for those who can work within its limitations, rapid injection molding may be suitable for producing some optical parts (and cost as much as 90 percent less than traditional tooling).

Continued on next page...
This part was molded in polycarbonate with F1 finish on one mold half, F0 finish on the other and texture around the rim. (F1 has most cutter marks polished out.) The mold was designed with an unbalanced edge gate, and 3,400 pieces were produced with no significant issues.

This part was molded in polycarbonate with A2 finish on both mold halves. (This is the highest polish but done by hand at Protomold.) The part has uniform, nominal thickness of .108” and was molded in a run of 750 pieces with no issues.

If you have questions about the suitability of the Protomold process for your optical parts, contact Protomold Customer Service Engineers at 877.479.3680.
To understand the importance of uniform wall thickness in rapid injection molding, imagine that the fluid injected into a mold is water rather than plastic resin. In a properly vented mold, the water, following the path of least resistance, will quickly and uniformly fill every nook and cranny regardless of the shape and size of the mold’s features. Resin, too, follows a path of least resistance through the mold, but in other respects resin is not at all like water. First, it is far more viscous than water and must be driven under pressure into the mold. Second, as it cools resin becomes viscous and eventually solidifies. And finally, after it solidifies—even after it feels cool to the touch—a molded plastic part can continue to shrink for hours. As a result, if a part is not properly designed a mold may be difficult or impossible to fill uniformly with resin.

In general, two types of features can cause thickness-related fill problems. The first is an “island” of aluminum, or core. This is a raised mold feature that forms a thin area in a plastic wall and through which the resin must flow. The second is an island of plastic, or cavity, a thick area of the wall that must be filled. Each represents a deviation from even wall thickness and presents its own challenges.

An example of an aluminum island is a decal recess in an otherwise uniform wall. If the recess reduces wall thickness significantly, resin will typically find a path of least resistance on either side of the recess. It will flow around the aluminum island and may fill the entire rest of the mold before backfilling the thin area. By then, the resin will have cooled, so the flow fronts that meet in that area may not fully meld, resulting in a cosmetically—possibly even functionally—compromised knit line. Also, if the thin area is not vented, it can form a “gas trap,” in which air is compressed by the advancing resin. As a gas is compressed it heats up in accordance with the ideal gas law (PV=nRT) and can “burn” the resin.

There is a simple way to create your decal recess without leaving a thin area in the wall; simply couple the recess on one side of the wall with a bulge on the other. In this way, the wall maintains its thickness and simply takes a slight detour. If the curve of the detour is smooth, the advancing resin will not slow, and the recess will fill right along with the surrounding wall area (see Figure 1).

Figure 1: Uniform wall thickness (1.1 and 1.3) allows the resin to flow freely when parts are being injection molded; whereas example 1.2 would cause a flow restriction of the material because of the thin area of the part.

Continued on next page...
An example of a plastic island would be a boss attached along its side to a wall. More resin at that location means more sink, typically on the opposite side of the supporting wall. The solution is to move the boss away from the wall, attaching it to the wall with a thin gusset (see Figure 2). To keep the gusset itself from acting as a thick area, it should be 50% +/- 10% of the thickness of the wall to which it connects. (The same applies to the walls of the boss in relation to the wall from which it rises.)

In addition to sinking, thick areas can be difficult to fill in the first place. Resin passing through a thin area to reach the thick one can cool enough to “freeze” before the thick area can fill, creating a void. Gate placement can help but may not provide a total solution. Avoidance of thick areas is a better answer.

The more challenging the design, the more resin choice matters. Polycarbonate, for example, will produce voids in virtually any thick-walled area. Polypropylene is prone to sink and the creation of tiny bubbles in thick areas. Acrylic, on the other hand, is generally better in thick cross sections. Glass-filled materials, the “bad boys” of the resin world, are almost always challenging, particularly in uneven geometries in which their impaired flow characteristics can create warp and a host of cosmetic problems.

In summary, there are three factors affecting resin flow: design, material, and gating. In general, you control the first two while Protomold controls the third. Protomold will typically gate at the thickest area and flow toward the thinnest. If you have questions about this (or anything else), contact our Customer Service Engineers at 877.479.3680. We'll use all the tools at our disposal, including Protolynx flow analysis software, to help you get the results you want.

To learn more about wall thickness related fill problems, view our video design tips on YouTube.
Round like a wheel, not like an egg

When making round parts—gears, pistons, plugs, and anything else that needs to fit, seal, or spin—start by asking yourself just how round it needs to be. If the answer is “as round as possible,” you probably need a center gate. The reason is that, in injection molding, resin fans out into the mold from the gate. With a center gate, that means an expanding flow front that stops at the outside edge of the mold where it meets the parting line once the mold is full. In Figure 1, because filling and cooling occur at roughly the same time around the radius, the part stays as round as possible.

If you’re designing a Frisbee®, which has featureless material at its center point, gate location can be easy, but what if there is a hole or feature at the center of your part that keeps you from placing a gate there? (See Figure 2.) If roundness is important, you may need to consider adding a dome (Figure 3) that can be machined off, or a plug in the hole (Figure 4) that can be drilled or machined out.

Figure 1: ProtoFlow® image—gating a part on center.

Figure 2: Cross section of the round part with a hole in the center.

Figure 3: Adding a round dome to machine off after molding.

Figure 4: Adding a plug to the center hold that will be drilled/milled out after molding.

Continued on next page…
The alternative to adding material at a challenging center point is to place your gate off center, resulting in an off-center fill pattern and inviting potential problems. If your part has a center core and you use an off-center gate, a knit line will form where the material flows to meet around the core (see Figure 5). That knit line will almost certainly cool and shrink differently from the rest of the material. Even without a core, an off-center gate means unequal flow length as resin moves toward the edges of your part. Uneven cooling can result in a slight egg shape in the resulting part—definitely not what you want in a spinning or precisely fitted part.

A center gate on a round part is particularly important with glass-reinforced resins. The radial fill allows the fiber filler to align in the outward direction of flow like the spokes on a wagon wheel. As the resin cools, this will offset the effect of transverse/perpendicular shrink, which can adversely affect flatness.

It is important to consider your ultimate goals and keep your options open when designing round parts. While adding material (like the domes or plugs mentioned earlier) to the part may require additional processes, the benefit of uniform radial fill will help make your part as strong, round, and balanced as possible and reduce the likelihood of problems. If you have questions about this tip, or other items regarding part design for injection molding, contact our Customer Service Engineers at 877.479.3680. We will be glad to discuss the function of your part and help with design considerations.


Check out our video design tip on knit lines on YouTube.
Resin performance can be measured in many ways, and many characteristics can be found in data sheets. One of the more difficult traits to reduce to numbers is finish, which often takes a backseat to more functional characteristics.

Many factors contribute to a finished part’s appearance—e.g., resin color or tendency to sink or “orange peel” (texture resembles the bumpy surface of the skin of an orange)—but the finish of the mold itself can be a key factor. The problem is that mold finish doesn’t always equal part finish. Resins like polycarbonate and ABS will typically match the mold finish quite faithfully. Conversely, the finish of glass filled nylon is more influenced by the glass content than by the mold surface, though most users consider its unattractive appearance a small price to pay for its high strength. Similarly, the thermoplastic elastomer Santoprene will show more-or-less the same matte finish no matter how you finish the mold, while providing flexibility and impact resistance that few other resins can match. Between these two extremes, resins like acetal and HDPE have both strengths and weaknesses when it comes to appearance and are reasonable choices when appearance is of some value but is not the primary consideration in resin selection. In short, if appearance counts, don’t just assume that mold finish will determine exactly what you get. If a specific finish is important, choose a resin that will accurately reproduce that finish.

To give some sense of how widely finish fidelity can vary, we’ve developed a new design aid we call Protogami. This consists of six linked tetrahedral pyramids that can be rotated through four different configurations.

Each tetrahedron shows four different finishes on a single resin:

- SPI-A2: Grade #2 Diamond Buff, 1-2 Ra
- PM-F0: Non-cosmetic—finish to Protomold discretion
- SPI-C1: 600 grit stone, 10-12 Ra
- PM-T2: Protomold texture, SPI-C1 followed by medium bead blast

Each of the Protogami’s four configurations shows six different resins in a single finish. The resins are:

- ABS
- Acetal
- Polycarbonate (PC)
- Thermoplastic elastomer (TPE)
- Glass-filled nylon (PA-GF)
- High-density polyethylene (HDPE)

In addition to showcasing resins and finishes, the Protogami highlights clip design, living hinges, the challenges of working in multiple resins, and design for ease of assembly. Request your own free Protogami, ready for assembly at www.protolabs.com/protogami.
Being transparent

After years of experience in mixing colorant into resin pellets, we have developed a new standard for the ratio of transparent colorant added to clear resins. This 1% ratio applies to all of the transparent colorants we stock, including:

- Trans Red UN3012TR
- Trans Amber UN2034TR
- Trans Green UN6881TR
- Trans Blue UN5016TR
- Trans Grey UN7903TR

(All of our stocked transparent materials are Omnicolor concentrates, manufactured by Clariant.)

Customers can, of course, provide their own pre-compounded transparent resins at any color ratio they choose, and current customers who are already having parts made using a different colorant ratio can continue to do so.

The decision to standardize on a 1% ratio for transparent colors was based on experimentation at varying ratios of colorant to clear base resin. The ideal colorant ratio turned out to be significantly lower than the 3% ratio Proto-mold recommends for coloring opaque resins.

Our tests showed that, at ratios higher than 1%, the resulting color is too dark, particularly in thick areas, and transparency suffers. At less than 1%, the mixing of colorant into the base can be too uneven, producing a marbleized or swirling effect. This occurs because colorant pellets are added to base resin pellets to produce a “salt and pepper” mixture in the hopper of the molding press. As pellets are heated and compressed in the barrel of the press, color from the dye pellets mixes with the melted base resin pellets. When processing a “salt and pepper” colorant mix at the press, the resin only goes through one extrusion process from hopper to part through the barrel.

Pre-compounded resins, where the pellets are mixed to a uniform color, go through multiple extrusion cycles to help make the color distribution more uniform. The fewer dye pellets per volume of base, the more dispersed they are and the greater the likelihood that some areas of base resin will receive little color before injection. At less than 1% color, this lack of wide dispersion starts to become clearly visible.

1/2%—Inconsistent color distribution throughout the part affects its appearance and may cause inconsistency from part to part even in the same run.

1%—Some swirling is common in “salt and pepper” colorant mixes, but we get a reasonably transparent part and reasonable consistency from part to part.

Continued on next page...
Note that even at our recommended 1% ratio, some color variation within and between parts can be expected. As suggested earlier, the best color consistency will always be achieved using customer-supplied pre-compounded resins.

To learn more about the “salt and pepper” process and colorant & color matching, read “What Color is Your Prototype?” on page 11.