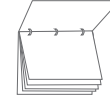


Print.

Format: Landscape

Page Size: Fit to page



Bind and save.

Bind in presentation
format for future reference

Design Tips

for Rapid Injection Molding

Volume 9

Real Parts. Really Fast.

www.protomold.com

protomold®

Design Tips Categorized by Topic

Page	TABLE OF CONTENTS	Material Selection	Design Guidelines	Quality Assurance	Understand the Process
3	Tips on Clips (Part 1)	●	●		
5	Tips on Clips (Part 2)		●	●	
7	Bigger and Better (Part Size Overview)		●	●	●
8	Cut to the Bone		●	●	
10	When NOT to Draft		●	●	●
11	It's Never Too Early for ProtoQuote®			●	●
13	Getting into Gears		●	●	●
14	Parts on a Budget: The Top Five Money-Saving Tips		●		●
16	Word Processing		●	●	●
17	Resin Selection — Check the Data Sheet	●		●	●
19	Sand Castles and Plastic Parts		●	●	
20	Bridging the Gap Between Prototypes and Production		●	●	●
22	Crush Ribs — Getting to the Point		●	●	●

Tips on Clips (Part 1)

One of the many benefits of plastic resin is the ease with which it can be molded into complex shapes. This often allows a single part to replace two or more parts made of other materials. Among the complex features that can be molded into a plastic part are a variety of integrated snap connectors, which can also eliminate the need for still more parts, such as screws, or for secondary processes like adhesive bonding.

The first consideration in snap connector design is material. In order for a snap connector to work, some area of the part must flex. This is why snap connectors can work in plastics (though not all plastics) but not in rigid materials like glass or ceramic. Resins that are especially suited for snap-fitted parts include ABS, polycarbonate, unfilled nylon, polypropylene, and other resins with similar properties.

The most familiar type of molded-in connector, the hooked cantilever clip (Figure 1), will be addressed in this design tip. Other connector types, including annular snap fits and torsional snap fits, will be addressed in Part 2 of this design tip next month.

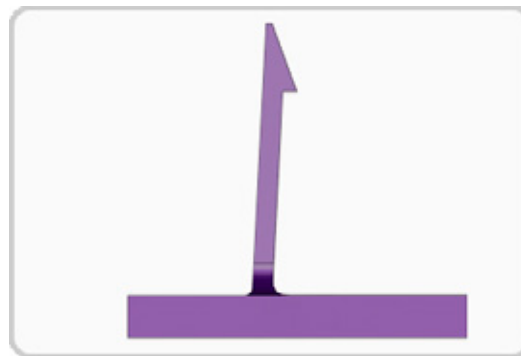


Figure 1: Cantilever clip with 90° hook face

Cantilever clips are used in a variety of applications (e.g., access panels in electrical devices) and can take many forms. Two key questions in designing such clips are:

- Do you want the connection to lock or to release with a pull?
- Do you want it to release at all or to be permanent?

If the clip's hook face is at 90° to the direction of connection, the connection will lock and cannot be undone by a simple pull (unless you pull hard enough to break the clip). If, however, the latching face of the hook is angled (Figure

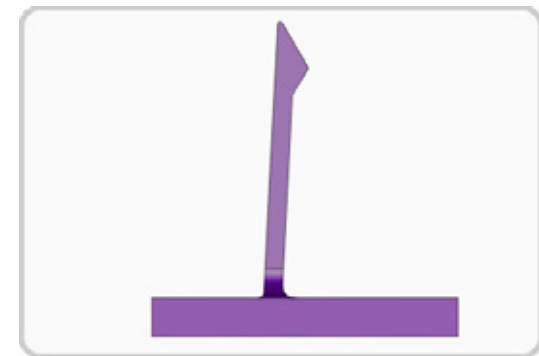


Figure 2: Cantilever clip with angled hook face to facilitate removal

2), a simple pull will release the connection. If you want a locking but non-permanent connection, say for an access panel, you can angle the hook face at 90° but allow the hook to be pushed manually out of its slot to unlock the connection. This is simple if the hook is positioned on the outside of the part. If the hook is located behind a wall, the designer can provide a "window" through which the hook can be accessed (Figure 3). Design of a cantilever clip determines its effectiveness and durability. The clip's arm must flex enough to allow it lock and unlock without breaking or deforming. This ability to

Continued on next page...

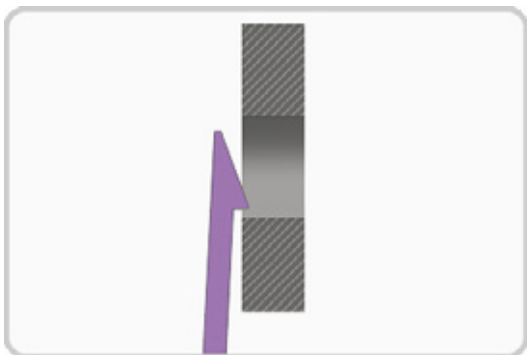


Figure 3: Clip positioned in a “window” to allow unlocking

flex depends on several factors including the material’s Young’s modulus, the angle through which the clip must deflect, determined by the depth of the hook, and the shape and length of the clip’s flexing arm. (Detailed formulae for clip design can be found at efunda.com.) They are also incorporated in many CAD programs, eliminating the need for separate calculation. Finite element programs can also be used to adjust the clip design to avoid breakage. Additional information on clip design can be found in a previous Protomold design tip.

Because the length of the clip’s flexing arm is critical and some designs offer limited space, there are several ways to increase the effective length of the arm.



Figure 4: Clip folded to increase the flexing arm’s effective length in limited space

- The arm can be folded into a U shape, as is often seen in battery compartment covers (Figure 4).
- The wall from which the arm extends can be notched, making that segment of the wall an extension of the arm.
- The wall from which the arm extends can itself be made flexible, reducing the amount by which the arm must flex.

Because a clip is, by its nature, designed to catch, it can, depending on its orientation, act as an undercut in a two-part mold. There are three ways to deal with this.

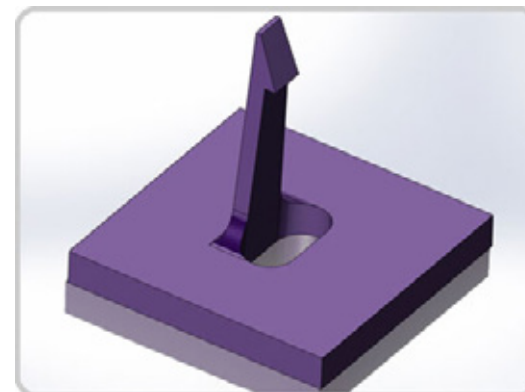


Figure 5: Clip with hole at the base to allow use of a simple two-part mold

- The simplest is to use a sliding shutoff extending through a hole at the base of the clip to form the bottom of the hook and one face of the flexing arm (Figure 5). This allows use of a simple two-part mold.
- A side-action cam can form the hook and then withdraw before the mold opens. This is an effective, but more complex approach.
- A pickout can be inserted manually into the mold to form the clip and then manually removed from the finished part and reinserted into the mold for the next cycle. Get more information on pickouts.

For further tips on clips, check out this Video Design Tip on Spring Clips: <http://bit.ly/Wu4vyM>.

Tips on Clips (Part 2)

Plastic's ability to flex without permanently deforming allows molded parts to incorporate a variety of snap fasteners other than the common hooked cantilever clip. These include:

- annular (round) snaps
- torsional snaps, which store return force by twisting rather than bending
- compressive snaps, which work by compressing and then returning to hold the fastener in place.

Annular Snaps

Annular snaps are used in a variety of everyday applications, from the tiny snaps that close the windflaps on jackets to larger ones that fasten caps to pens to the still larger lids on plastic yogurt containers. While these are all designed for ease of opening and closing, annular snaps are also used, with slight modification, in “childproof caps,” which can be easily opened in one position but are virtually impossible to open in any other. And while many applications support repeated opening and closing, annular snaps can be used in industrial applications for permanent fastening, typically ensured by the angle of the locking edges of the snap.

Whatever their application, annular snaps operate by elongation and recovery, typically of the female component. This restricts the materials that can be used to those with relatively high elastic deflection limits, the point where material fails to fully recover from

deformation. Maximum permissible strain varies for different materials, from about 50% of the strain at break for most reinforced plastics to more than 70% of strain at break for more elastic polymers. Detailed information on annular snap fit design can be found at <http://bit.ly/Wk06E2>.

Torsional Snaps

Torsional snaps store closing force when opened by imparting twist to a torsion bar at the pivot point of the arm, as opposed to bending the flexing arm as in a cantilever clip. But in most other respects—hook design, etc.—the torsional snap is similar to the cantilever clip. Rocker clips are typical torsional snaps. For detailed information on the design of torsional snap fits, visit <http://bit.ly/YDpP6E>.

Compression Snaps

Compression (or interference) snap fits can take many forms. The compound dovetail snap used in the Protomold sample design cube (Figure 1) is one example. This clip is dovetailed in two directions to lock together two of the cube's six faces. In one direction, the male component is highly tapered providing an unbreakable connection between the two components. In the other direction, the male connector is only slightly tapered providing a connection that can be easily undone to disassemble the cube. As the male connector is pushed into position, material in both male and female components is compressed and then released as the snap moves into its locked position.



Figure 1: Clip is dovetailed in two directions to lock together two sides of a box.

Another example of a compression snap fit is found in the Reptangles™ building system. The challenge here was to design a connector that would mate when the parts came together in any direction within a 90° arc. The patented connector (Figure 2) consists of a triangular arch that fits into a corresponding slot. “Fingers” in the slot walls grasp the hollow under the arch to link the pieces firmly together while still allowing easy disassembly.



Figure 2: Arch-shaped connectors fit into a slot in an interference fit.

Continued on next page...

A compression snap fit can create moldability challenges. In some cases the connector can be formed as a “bump-off,” in which the part is slightly undercut but the material flexes to allow ejection. In the case of Reptangles, bump-offs were not required, as both arch and slot components of the connectors were formed by sliding shutoffs passing through holes in the part to form the undersides of protruding features.

The wide variety of clip and snap options allows designers a great deal of latitude in creating integrated connectors.

Bigger and Better (Part Size Overview)

We've posted tips in the past regarding maximum part size, but since our moldable dimensions have continued to increase, we thought it was time for an update. Some of the figures that follow are simple and absolute; others may vary based on multiple factors. If your design seems to be approaching the limits, the best way to determine whether we can mold it is to submit a 3D CAD model for a free online ProtoQuote®. Knowing the guidelines in advance, however, can help speed up the process.

Volume

The simplest and most absolute limit to part size is the total volume of your part, which cannot exceed 59 cubic inches (967 cc). That is the maximum capacity of the barrel from which our largest presses inject resin into a mold. This maximum figure has increased significantly over time as we've added larger-capacity presses to our production floor, and the good news is that if you follow standard guidelines for wall thickness, 59 cubic inches of resin goes a long way.

Height

This one is also simple: the Protomold process allows parts to be milled to a maximum depth of four inches from the parting line. So, if the parting line falls exactly halfway between top and bottom, total part height can be up to eight inches.

Mold Area at the Parting Line

Because resin is injected into a mold under pressure, the two halves of a mold must be clamped together during injection to keep them from separating prematurely. The pressure that must be overcome equals injection pressure in psi (pounds/in²) times projected part area (cross section of the part, in square inches, at the parting line). Our presses can exert up to 600 tons or 1.2 million pounds of clamp pressure, which allows a projected part area at the parting line of up to 175 square inches.

Outline

Pressure generated by injection is exerted in all directions and does more than try to force open the mold. It also presses outward against the sides of the mold. If that force is exerted over a large enough area and the mold wall is too thin, the pressure of injection can actually bow the walls of the mold. To prevent this, as the part is milled deeper into the mold (increasing the area of mold wall exposed to pressure) the mold wall must be thicker. Thicker mold walls reduce the maximum rectangular outline into which the part must fit. In other words, the taller the part from the parting line, the smaller the maximum outline (as defined in Table 1).

Maximum Part Height	Maximum Rectangular Outline
shallow	29.6" x 18.9" (751 mm x 480 mm)
1 inch	27.6" x 16.9" (701 mm x 429 mm)
2 inches	25.6" x 14.9" (650 mm x 378 mm)
3 inches	23.6" x 12.9" (599 mm x 327 mm)
4 inches	21.6" x 10.9" (549 mm x 277 mm)

Table 1

Allowance for Cams

Because side action cams must fit within the allowable mold outline, they also reduce the maximum size of a part. The space a cam requires will vary with the stroke required to create the undercut, but it can be significant. The Protomold software will make appropriate allowances for side actions, but designers should be aware that parts with side actions must fit within a reduced outline.

Part Wall Thickness

Larger parts generally mean longer resin flow paths, which, in turn, require thicker walls; how thick depends on the resin. Minimum wall thickness for a long-fiber-filled resin can be nearly 2.5 times that for a nylon wall. Check out our information on range of thickness for various resins.

Cut to the Bone

We've all seen cartoons of the caveman with his waist-high stone wheel. It literally weighs a ton and never seems to be attached to anything. We've seen images of chest-high, solid wood wheels on a peasant's lumbering cart; better than nothing but still far from race-ready. The real breakthrough in wheel design came when some genius realized that a wheel needs a hub and a rim but that most everything in between is superfluous. It was only with the invention of spokes that the wheel—suddenly lighter, more economical, and with less angular momentum to spin up and brake down—came into its own.

The same principle applies to plastic parts. Plastic is inherently light, generally economical, and relatively easy to form. But that doesn't mean that it can't, with thoughtful design, be lighter still and even more economical, all without sacrificing performance or complicating manufacturing. Accomplishing this requires the elimination of unnecessary material, which raises two questions: why and how.

Why has several answers, and the first is cost. A recent item in *The Plastic Exchange* pointed out that buyers were snapping up supplies of resin in late January 2012 to avoid price increases expected in February. The impending increases ranged from 02¢ to 06¢ per pound on some relatively inexpensive resins: polyethylene and polypropylene. If pennies-per-pound savings are that motivational, any significant reduction in the per-part volume of resin could be even

more worthwhile, and such savings would be proportionally greater on more expensive resins.

The second reason to trim down is weight. From bicycles and cars to military gear and aerospace, weight reduction has become a kind of Holy Grail. Plastic is, of course, lighter than metal, but less plastic is lighter still. As the saying goes, "You can never be too thin or too rich," and in today's market being lighter can make you richer by reducing cost and increasing functionality.

Another argument for trimming is style. Perhaps because skeletonization is associated with high-tech, sport, and the like, it has developed a certain cachet. Minimalism sells, which brings us to the next question, which is, "**How?**"

One of the simplest ways to lose weight in a plastic part is to core out thick areas. This not only saves money and weight, but also prevents problems like sink, voids, and warps. Another approach is skeletonization. The concept has been used for years and applied to all sorts of materials. Wood and metal trusses are designed to maintain strength while minimizing material requirements. So is a honeycomb. In many cases, hollow cylinders can replace solid posts, and properly positioned arches can redirect forces and replace bulky solid supports. All of these techniques can be incorporated into plastic parts. The challenge is to remove material without impairing function. Figure 1 shows an example of the process.

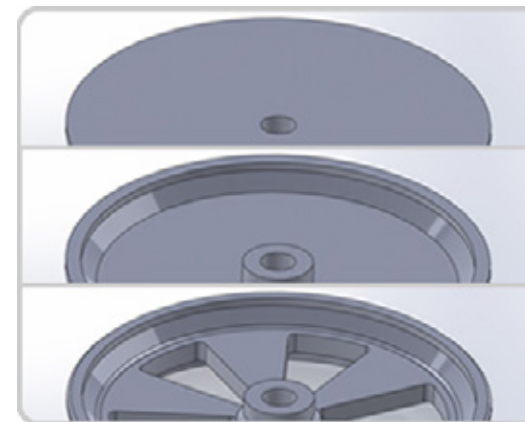


Figure 1: This part evolved from a solid disk to one with cored-out sections to a fully-skeletonized spoke wheel with no loss of functionality.

Since resins vary widely in their characteristics, the amount of material that can be eliminated from a particular design will depend on the material, and choices of material and form will interact as the designer searches for an optimal design. Finite element analysis (FEA) software is great during the early virtual phase of development, but prototypes using actual materials are necessary for testing and refining the design as the process moves forward.

Thoughtful prototyping by rapid injection molding can provide maximum information at minimum cost. You can test multiple resins in the same mold (as long as you remember that resins with different shrink rates will produce parts of slightly different dimensions). Depending on what you plan to test, e.g., individual part performance rather than a final assembly, this may not be a problem. Also, if you plan to test varying degrees of skeletonization, remember that metal is easy to remove from a mold but

hard to add, and less metal means more plastic. In other words, start with the most trimmed down version of your part. If it doesn't stand up to testing, you can add plastic for your next iteration by milling more metal from the mold instead of starting from scratch. As with skeletonizing a part, it's a way of trimming fat from your development process, maintaining effectiveness while reducing cost and effort.

When NOT to Draft

Plastic part design can be a complex process under any circumstances, and when you are designing entirely new types of product the challenges can be greater still. In design you rarely know what little trick or idea is going to make the difficult easy or the seemingly impossible possible; that's why we offer new design tips every month. That said, every once in a while we run across one that seems so important that it's worth revisiting. Here is a "Design Tip Classic" from December 2009...

We've talked so often about the need for draft in injection molded parts that you may be shocked to hear that there are features that not only don't need to be drafted, but that work better if they aren't drafted. It's true, and the reason we haven't mentioned it before is this is a new capability added to our injection molding process.

The features in question are typically screw holes used to connect plastic parts—front and back halves of a plastic shell, for example—with thread-forming screws. The holes are formed by posts in the mold called “cores” (see Figure 1).

Previously, we created cores in the mold by directly milling them from the aluminum mold body. Tall thin cores could “stick” to the plastic part and break off when the parts were ejected. To strengthen these cores and reduce ejection stresses we required them to be drafted as tall narrow cones. The resulting tapered screw holes could be problematic. Unless the screw was also tapered (like a wood screw), it would become

tighter as it was screwed into the tapered hole. If it got too tight, it would crack the part. If it was too loose it could “strip” and fail to hold.

Long, straight screws, tapered pilot holes, and knit lines were a bad combination. If the hole and corresponding screw length were short, the part could be safely produced, but designs with deeper holes unfortunately went back to designers as no-quoted parts.

That's all in the past. Now, Protomold can produce high-aspect-ratio small diameter holes using steel core pins in the mold. Say, for example, you are designing a part with a 3/4" deep, 1/8" diameter hole (see Figure 2). You simply include that feature in your 3D CAD model; our proprietary software will go to work and design the mold with a cylindrical steel core pin for forming the hole (see Figure 3).

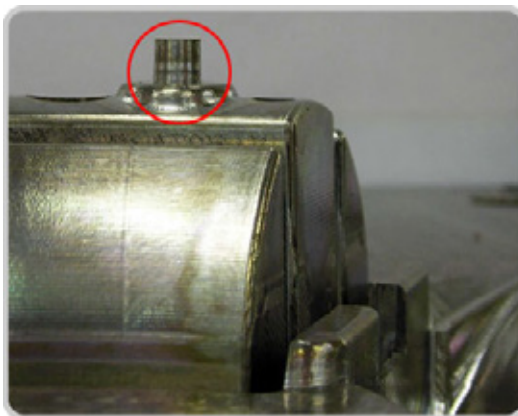


Figure 1: Example of a core used to form a hole in a molded part.

This innovation changes two things for you. First, we can now mold parts with deeper, narrower holes. Second—and here's the shocker—you don't have to draft those features. The reasons are simple. A steel pin is strong enough to handle the stress of ejection and its surface is smooth enough to release cleanly from the part without draft. And, while there shouldn't be any cosmetic effect on the resulting part, if there is, it will be inside the hole where it won't be seen.

The size of the hole in your part will be determined by the size of the thread-forming screw you'll use for assembly. The hole itself will be slightly larger than the minor diameter of the fastener—the diameter of the shaft at the root of the threads. Typically, the manufacturer will specify a diameter for the pilot hole in their screw specs. Finally, note that some screws will be specified for particular plastic resins, so if you change your resin during prototyping you should make sure you're still looking at the right type of fastener.

It's Never Too Early for ProtoQuote®

By this time, you probably know all about ProtoQuote interactive quotes: they're free, automated quotes for rapid injection molding that allow you to interact with the material choices, quantities, and design feedback (see Figure 1). The design analysis suggests where and how your design should be tweaked to improve moldability. For most users, ProtoQuote fits into the design process somewhere after napkin sketches and 3D printing (or other additive prototyping processes) and before the injection molding of production parts. That's the tried and true methodology, but we'd like to suggest an alternative approach that you might find useful.

On your next project, consider uploading your 3D CAD model to get a quote from Protomold before you make your first prototype no matter what prototyping process you plan to use. In other words, whether you start with an injection molded or machined prototype from Proto Labs, a layered prototype from an outside service bureau, or a layered prototype made on your own in-house equipment, get a ProtoQuote first.

The reason is simple. If you plan to use injection molding for final production, it is crucial to know your design is moldable, and the earlier in the development process you know this the better. As we've often pointed out, Protomold injection molded prototypes will tell you whether your design is moldable. (There are, of course, other reasons to use rapid injection molding in your development process—it provides functionality



Figure 1: Dropdown menus allow you to change materials, A- and B-side mold finishes, and quantities to see the impact on price.

information that you can't get from any of the additive processes—but we understand that the additive prototyping processes can be useful in early stages of development.) What we're suggesting is that you not wait until you're ready for molded prototypes to get the design analysis that comes with your ProtoQuote. We'd even go so far as to suggest that you request a ProtoQuote even if you have no intention of ever getting a molded prototype before committing to production tooling!

The benefit for you is that our quoting software can identify potential moldability issues (see Figure 2), like missing draft or potential for sink, at any point in development. All you need is a 3D CAD model you can upload, and since all of the above processes require a 3D CAD model, requesting a ProtoQuote is fast, simple, and

free. In return you get valuable information that could potentially prevent molding problems or the need to "go back to the drawing board," later. If your part has undercuts, ProtoQuote will find them. Walls that are too thick or too thin for effective molding? Additive processes won't find them, but our proprietary quoting software with design analysis will.



Figure 2: The ProtoView® 3D viewer allow users to view a fully rotatable model of the design and zoom in on potential moldability issues.

If you plan to use ProtoQuote this way, there are a few things to keep in mind. Along with your 3D CAD model, the software will optionally ask for your resin choice, the number of parts you want molded, and your preferred speed of delivery. Your model will then be run through the largest compute cluster in the industry, and you

Continued on next page...

will receive, in less than 24 hours, an interactive price quote along with your design analysis. If you aren't ready to order molded prototypes, you can ignore the quote and go straight to the analysis. Color-coded markups on your CAD model will indicate potential moldability issues (see Figure 3). If you will be using the Protomold injection molding service, these concerns will have to be addressed. If, on the other hand, you are planning to make prototypes using any of the additive—layered—prototyping processes, you don't have to make these changes, but if you plan to use injection molding for production you'll probably want to address them anyway.



Figure 3: The grey area on the rim of the part indicates an area with zero draft, which may cause issues such as drag marks, or distortion from ejection stresses. This is one of the many design considerations for optimal performance in the injection molding process.

Once you have made the indicated changes in your 3D CAD model, you can resubmit it at protomold.com to make sure you've addressed everything. As development progresses you'll probably want to make one or more injection molded prototypes to use for functional testing, because real prototypes provide information that CAD models, simulation software, layered prototypes, and even a ProtoQuote design analysis can't.

Watch short videos about ProtoQuote and Proto Flow (simulates resin flow, predicts pressure, and anticipates resin fill issues) at <http://www.youtube.com/user/ProtoLabsInc>

Getting into Gears

As design tools and techniques become more sophisticated and the arsenal of available resins grows, plastic continues to replace other materials in a variety of applications. Gears are among the mechanical parts being made increasingly of plastic. To effectively replace metal, the design and material must, of course, be suited to the demands of the application, but moldability of the part is equally important. All injection molding processes have specific capabilities and limitations, and if you plan to have Protomold produce gears it is crucial to understand the requirements of our molding process.

The primary challenge in the rapid injection molding of gears is the teeth. In order to be able to mold a gear we have to be able to fully fit an end mill into the area of the mold that will form a gear tooth. Figure 1.1 shows what happens when a gear tooth is too small for our smallest end mill. The mill cannot reach the end of the tapered tooth, so this gear cannot be produced using our process. Figure 1.2 shows a gear with teeth large enough to accommodate our end mill.

Note that while the mill can reach the end of the tooth in Figure 1.2, it cannot reach into the corners of the squared off tooth. Teeth on this part will need to have more rounded corners. When you submit such a part for a ProtoQuote®, the inability to produce the squared outside

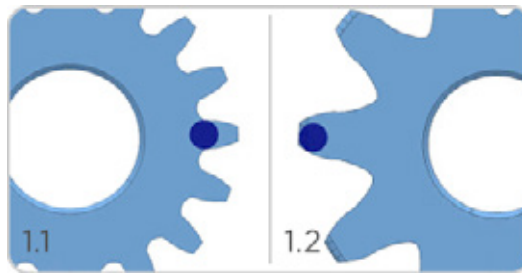


Figure 1: The dark blue dot represents the end mill that would be used to form a gear tooth in the mold.

corners will be noted in the design analysis, and you will have the opportunity to decide whether to modify your design accordingly. Depending on your design, this rounding could affect the engagement of gear teeth causing unacceptable “slop” in the operation of the gears. Your 3D CAD software can help you determine whether the modification will cause problems.

Another design consideration for gears is the side-to-side width of the gear. Wider gears may be needed to handle greater force, and the width of the gear determines the depth to which a mold will have to be milled. Deeper cuts require longer mills, and because long thin mills could wobble during cutting operations, our minimum mill radius increases with cut depth. If cut depth is great enough we use tapered mills to prevent wobble during cutting. As a result, very deep cuts require draft (see Figure 2).

Minimum Endmill Radius	Maximum Cut Depth	Draft Required
.010"	.120"	none
.013"	.190"	none
.013"	.310"	.5°
.013"	.420"	1°

Figure 2: Requirements affecting the radii of outside corners.

When draft is required on gear teeth, it's to allow for proper mold milling, not ease of ejection. While the amount of draft required for wider gears is small, as in the case with rounded gear teeth drafted gear teeth can affect the engagement of gears. In some cases, designing complementary draft into the teeth of mating gears can eliminate engagement problems.

Because they have to move and mate, most gears are made from a relatively small range of high-lubricity resins. For this reason, our ability to produce gears is not generally affected by resin choice. Following the above guidelines should allow you to design gears that can be effectively molded using our process. If moldability problems are found, they will be highlighted in your ProtoQuote design analysis.

Parts on a Budget: 5 Money-Saving Tips

We are sometimes asked, “How can I get my parts made for less money?” This design tip shows you the best ways to cut costs from your Protomold project. A word of warning: most of the ways to save involve changing the design of your part. Obviously, if your part needs to be a certain way, it needs to be that way. On the other hand, if you have flexibility in your design constraints, there are many ways to save.

As you consider these tips, you need to keep your end use of the parts in mind. Two major constituent groups use Proto Labs: people who need prototype parts and people who need short-run or bridge production parts. Within the prototyping constituency, there are subgroups interested in fit, appearance evaluation, process validation, strength testing, and so on. Saving money by eliminating a cosmetic finish might be just fine for a prototype part used for strength testing, while it would not make sense for a prototype case used in a marketing sample. For prototypes, minimizing the tooling cost usually makes the most sense. For people who need production parts, it might make sense to incorporate ideas to reduce per-part costs even if the cost of the tool increases as a result.

In no particular order, here are the five money-saving tips we most often give our customers:

1. Design all the undercuts out of your part.

We have to charge more to add side action cams, bump-offs, and pick-outs to your tool

to create undercuts. In the case of pick-outs, we also charge a per-part fee to keep a person standing at the press inserting pick-outs into the tool before each shot and then removing them from each part after molding (see Figure 1).

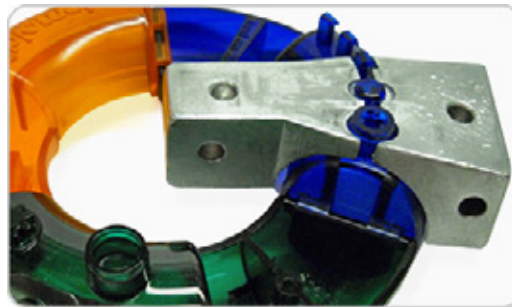


Figure 1: The two-piece pick-out used to create the features of this part needs to be manually removed from the part after ejection, and therefore adds cost.

2. Make the part smaller. If you can make the part smaller, there are multiple cost benefits: smaller parts fit in smaller tool bases, saving tool material cost; smaller parts have smaller cavities and therefore less milling time is needed to create the cavity; smaller parts use less material; and so on.

3. Take details out of the part, especially small or fine detail. Protomold supports features as small as .020" [.50mm]. Fine detail is created using tiny end mills. Tiny stubby end mills cut very slowly, and longer tiny end mills, needed to reach fine detail deeper in the part, cut even slower. You may reduce milling time by keeping

your geometry thicker than .050" [1.27mm]. Using our free ProtoQuote® design analysis allows you to compare the cost of a more complex geometry to a modified cost-focused geometry.

Text is very time-consuming to mill, especially text recessed into the part. Logos are similar to text, and avoiding them will also reduce milling cost. Sometimes EDM electrodes are suggested instead of milling for small text features in tools—this is not a cost-reduction strategy since the electrodes must also be milled and the EDM process requires an extra step.

Another detail that can add cost is ribs. Ribs require slots to be milled into the tool, and thinner and deeper slots need smaller and longer end mills, which cut more slowly. If smaller ribs can be combined and fewer larger ribs can be used instead, the tool cost is usually reduced. If the objective of ribs is to core the part to reduce production material usage, AND you care less about per-part cost than up-front tooling cost, AND you can live with possible sink or warp, then you may decide to make a thicker solid part and eliminate some ribs altogether.

One of our Customer Service Engineers sums up the detail-elimination technique as: “Less waffle, more pancake.” Of course, if what you’re making is a waffle iron...

Continued on next page...

4. Use lower grade finishes for your part.

Finishes are applied to our tools by hand, an expensive process. If your part is not cosmetic, don't use a costly high-cosmetic finish. If the end use of the part is cosmetic, but the immediate use is non-cosmetic (for example, a bezel initially used for fitment or ergonomic tests), select a lower-end finish. We can always add a better finish to the tool after your design is finalized. If you don't know what the different finishes look like, check your Protomold design cube: all of the standard Protomold finishes are on the cube. If you don't have a design cube, your sales person or customer service rep can send you one. We also have sample plaques showing what different finishes look like in different standard materials (see Figure 2).



Figure 2: Our material sample plaques show seven different finishes offered by Protomold. Talk to your sales rep if you are interested in receiving samples.

5. Consider secondary operations. Surprisingly, depending on the quantity required, you may find it faster or more cost-effective to use a secondary operation to complete your parts rather than having the part molded complete. For example, you could drill a hole after molding rather than use a side action or pick-out. Another example might be to cut an internal undercut in a secondary operation rather than a pick-out. Sometimes you can create a “parent” part and then machine it to form different “children,” or variants, so you can use a single mold to create a product line. You might be able to machine in a bump-off feature rather than have an undercut mold made. Please note that Protomold does not do secondary operations.

Word Processing

There are lots of reasons to add text to a part. It could be an assembly instruction, a part number, a legally-advisable warning, or simply a logo (see Figure 1). Whatever the reason, text characters tend to be the smallest features of a part and, as such, deserve the designer's careful attention.



Figure 1: Off/On switch instructions on a part.

The first thing to keep in mind is that it works much better if text on a plastic part is raised above, rather than recessed into the part (which means it will be milled into the mold). Raised letters on a part are easier to read, and recessed text in a mold allows for polishing, whereas raised letters in a mold make it difficult to achieve a good finish.

The second issue is consistency of wall size in your lettering. Avoid serif fonts, the ones with the little squiggles at the ends of uprights. The serifs are typically narrower than the primary lines of the letter itself, making them too small to mill. Instead, use a sans-serif (non-serif) font

like Century Gothic Bold (the default font in SolidWorks). Other common sans-serif fonts are Arial and Verdana. In general, remember that while most 3D CAD programs allow you to use standard Windows fonts, you should resist the temptation to get cute without a good reason.

The third issue is the size of the letters themselves (see Figure 2). Text doesn't need to stand very tall above the surface of a part — .02 inches is plenty — but even so, the rules for thin ribs apply. You don't need to measure the thickness of every line of each letter; just stick to font sizes of 20 points or more and use the Bold version of the font, and odds are excellent it can be milled (see Figure 3). In some cases, we can mill smaller fonts. If you need to do so, submit the part with the smaller text for a ProtoQuote® interactive quote and the quote will show any required changes or advisories. You can also contact a Proto Labs Customer Service Engineer at 763.479.3680 to discuss your project.



Figure 2: The “o” is too small to mill.

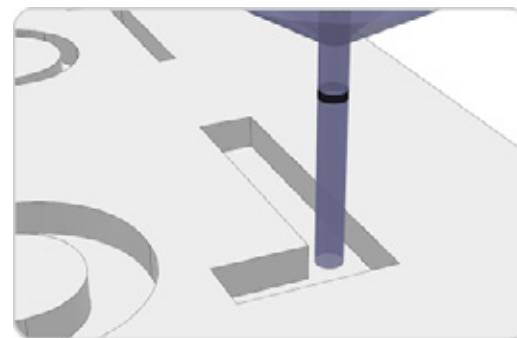


Figure 3: The “L” is a larger font size and allows room for the tool to mill.

Finally, if text is located at the top of a tall feature — a tall rib, for example — the text may have to be larger. In summary, for best results when incorporating text:

- Design your parts with raised text.
- Use a bold sans-serif font.
- Size text to 20 point type or larger.
- Stay away from the tops of tall features.

If you are wondering whether you've designed your text properly, simply upload your 3D CAD model for a free ProtoQuote®. If there are any problems you'll know by the next day.

Resin Selection — Check the Data Sheet

Whether you realize it or not, when you design a molded plastic part you are working at two levels. On the macro level you are creating a shape—simple or complex—that meets a variety of needs. At the same time, you are designing at the molecular level by your choice of resin or resin blend. Some aspects of a part's performance are determined primarily by shape. Others, such as strength, are influenced by both shape and material. And still others, like lubricity, are determined by material alone.

With thousands of resins and blends to choose from, material selection can seem daunting. Fortunately, there are some excellent resources that can help simplify the process. General information on a handful of the most commonly used resins can be found on our Protomold design guidelines page. But for more detailed information on literally tens of thousands of resins, the go-to source is the Prospector Plastic Materials Database at **UL IDES**. Here, at no charge (after free registration) you can find detailed data sheets on most of the world's resins (see Figure 1). Data is provided to UL IDES by material manufacturers and updated weekly. You can register and create a personal Prospector account by going to www.ides.com/protomold.

Let's start by reviewing this **sample data sheet** for a clear Polycarbonate material, Makrolon-2458. It begins with a general description of the resin and its application, in this case medical devices. The feature list includes characteristics needed for medical application including biocompatibility

Makrolon® 2458 Data Sheet		UL IDES Prospector	
Polycarbonate		Already have an account? Login Here	
Supplied by Bayer MaterialScience LLC		View Complete Data Sheet	
		Search 85000+ Data Sheets	
		Create Free Account	
Home > Plastics > Makrolon® > Makrolon® 2458 Data Sheet			
Product Description			
MVR (300 °C/1.2 kg) 19 cm³/10 min; medical devices; suitable for ETO and steam sterilization at 121 °C; biocompatible according to many ISO 10993-1 test requirements; low viscosity; easy release; injection molding - melt temperature 280 - 320 °C; available in transparent and opaque colors			
General			
Material Status	• Commercial: Active		
Availability	• North America		
Features	• Biocompatible	• Good Mold Release	• Steam Sterilizable
	• Ethylene Oxide Sterilizable	• Low Viscosity	
Uses	• Medical/Healthcare Applications		
Agency Ratings	• UL 94 V-0	• UL 94 V-1	• UL 94 V-2
	• UL 94 V-0	• UL 94 V-1	• UL 94 V-2
RoHS Compliance	• RoHS Compliant		
Appearance	• Clear/Transparent	• Colors Available	• Opaque
Processing Method	• Injection Molding		
Multi-Point Data	• Creep Modulus vs. Time (ISO 11403-1)	• Secant Modulus vs. Strain (ISO 11403-1)	• Viscosity vs. Shear Rate (ISO 11403-2)
	• Isochronous Stress vs. Strain (ISO 11403-1)	• Shear Modulus vs. Temperature (ISO 11403-1)	
	• Isothermal Stress vs. Strain (ISO 11403-1)	• Specific Volume vs Temperature (ISO 11403-2)	

Figure 1: Screen shot of material data sheet from Prospector Plastic Materials Datasheet at UL IDES.

and sterilizability. This particular resin is also characterized by good mold release, which could be critical for a part in which draft must be minimized, but may affect your ability to paint or plate the part. The agency ratings and RoHS compliance concern environmental and safety regulations are also available on each form. And appearance, of course, relates to cosmetic issues.

The remainder of the data sheet provides detailed information on the resin's physical, mechanical, thermal and other characteristics. These may all be relevant to the performance of your finished parts, but one in particular could be important if you plan to compare prototypes in more than one resin. Molding shrinkage (see Figure 2) helps Proto Labs determine how much

Physical
Density
Apparent Density
Melt Mass-Flow Rate (MFR) (300°C/1.2 kg)
Melt Volume-Flow Rate (MVR) (300°C/1.2 kg)
Molding Shrinkage
Across Flow
Flow
Across Flow: 0.0787 in ²
Flow: 0.0787 in ²
Water Absorption

Figure 3: Molding shrinkage specifications are listed in the Physical section of material data sheets.

to adjust mold size to so that cooled, molded parts will match your model. If you plan to test multiple resins, you may want to choose resins with similar shrinkage ratings to ensure that the parts made from different resin in the same mold cool to similar sizes. ABS and polycarbonate, for example, typically have similar shrink rates, while those of ABS and nylon are very different. When shrink rates are different, consider the order in which you have them molded; reduce your cost by having a single mold retooled instead of making multiple molds in different sizes. When shrink rates are significantly different consider making 2 molds to ensure the parts are comparable in size.

Keep in mind that data sheets are only useful once you know which resins to consider. On the other hand, if you know the characteristics you want but don't know which resins meet those requirements, you can use IDES Properties

Search, a fee-based premium service. Properties Search lets you search by more than 500 properties, specifying the ranges you require and narrowing the list of potential resins to a workable number.

If you have questions about reading data sheets, you can contact your Account Manager or a Customer Service Engineer at 877.479.3680. They can answer questions regarding properties, but they cannot recommend specific resins. For recommendations on specific resins, a custom compounding specialist like Polyone or RTP should be able to help you. Keep in mind, however, that while outside resources can help with information about resins, no one knows your application as well as you do, so the final choice of resins is yours. The good news is that testing resins in the prototyping phase can help you determine whether you've made the right choice or need to expand your search.

Sand Castles and Plastic Parts

Ever wonder why the little plastic buckets kids use at the beach have tapered sides? Hint: it isn't for hauling water. It's for that other popular beach pastime of the pre-teen set: making sand castles. Moist sand is pretty fragile stuff and the surface tension created by the water between the grains is just barely strong enough to turn a bunch of particles into a temporary solid. A good shaking would pull them apart, and since the same surface tension that sticks sand grains to one another also draws them to the sides of the bucket, a good shaking is what it would take to get damp sand out a straight-sided bucket (see Figure 1). The result would be a pile of loose damp sand and a powerful lesson on the importance of draft for the young castle-builder.

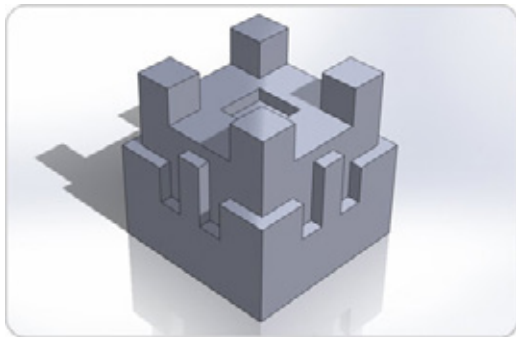


Figure 1: Because the surfaces that are in contact with the sand are undrafted, surface tension will cause the sand castle to fall apart upon lifting the mold. Similarly to prevent damage or drag marks on the part, surfaces that are parallel to the line of a mold opening may have to be drafted—angled away from the line along which the part will be ejected.

In the more typical tapered bucket, however, as the mass of sand begins to move out of the bucket it also pulls directly away from the bucket wall. Less contact means less surface tension between the sand and the bucket (essentially, less friction), and less friction means no need for shaking; a simple tap is all it takes, and gravity does the rest. This mode of ejection may leave a few grains stuck to the plastic, but it also leaves the turret of your sand castle intact (see Figure 2).

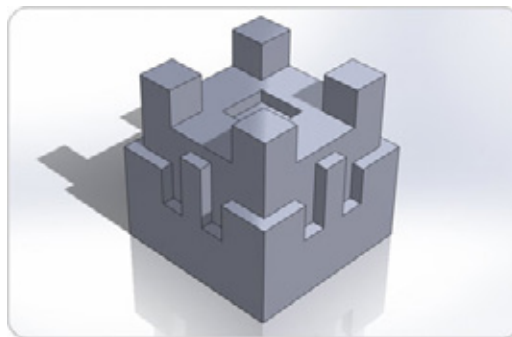


Figure 2: Draft facilitates the removal of the part from the mold and is particularly important where the molds are straight pull only, similar to this sand castle mold. Using at least 0.5 degrees of draft on all “vertical” sides eases release of your part, or, in this example, sand castle*.

The same principle applies to the design of plastic parts for injection molding. Molded plastic may be more solid than damp beach sand, but if the plastic gets dragged during ejection it's no match for a metal mold wall, and while the resulting part may hold its shape it won't hold its finish. So as long as your parts have surfaces that are more-or-less parallel to the direction of mold

opening you'll want to draft those surfaces no matter how complex or simple they might be.

One obvious example of such a part is the same plastic beach bucket we've been discussing. The bucket that acts as a mold for sand is itself a molded part. Its outside is formed by the A-side mold half, and its inside is formed by the mold's B-side. Shrinkage causes the cooling plastic to pull away from the outside mold half making that part of mold opening easy, unless, of course, the surface of the bucket is textured. Texture can act like tiny undercuts. To prevent damage to the surface, increase the draft. Light texture (PM-T1) requires three degrees of draft; medium texture (PM-T2) requires at least five.

At the same time, shrinkage makes the bucket cling to the B-side mold half from which it needs to be separated by the action of ejection pins. It is here that significant draft (and avoidance of texture) helps ejection and allows the injection molder more flexibility to improve the quality of the parts due to reducing difficulty of ejection.

The bucket we've described so far is essentially a cup. If we want to attach a handle we may want to add features that protrude outward from the surface or add holes in the sides of the bucket. To do so we might want to use side actions, which must be drafted parallel to the direction of cam opening (read more about side actions).

*NOTE: Sand may need 5+ degrees of draft where plastic may only need ½–2 degrees.

Bridging the Gap between Prototypes and Production

In the traditional approach to product development, there is a sharp line between development and production. Development begins with a light bulb over someone's head, proceeds through napkin sketches and CAD models, and ends, ultimately, with prototypes. At one or more points in the development process there may be input from the market, be it someone's best guesses, one or more focus groups, or actual market tests. And from start to finish there is always pressure to "get on with it," either because you need to catch up with a market leader or because you are the leader and someone may be catching up with you. But then, when you have reached your goal—a fully developed, marketable product—everything comes to a screeching halt and the drawings and/or models disappear into the "production machine," from which, weeks or months later, a whole lot of deliverable product appears and the rush begins again as it heads off to market.

In plastic molding, as in most other technologies, some aspects of this transition are unavoidable. Production molds are costly, and they take time to manufacture. It would be risky to begin producing them before the design had been fully proven in development, when even a small change could turn tens of thousands of dollars' worth of molds into doorstops and boat anchors. Traditionally this has always presented manufacturers with a dilemma. They could keep



Tensys Medical used Protomold for prototyping and pilot production on this T-Line® Tensymeter component, which allowed Tensys design engineers to significantly shorten the product design and development program time cycle.

development and manufacturing sequential and live with the resulting delay. Or they could treat them in parallel, starting on production molds before the end of development, cutting their time to market but running the risk of having to go back and start tool-making over again. It was a painful choice, because today's competitive global markets reward both speed and low cost. Manufacturers already recognize that rapid injection molding as a prototyping method can reduce both cost and delay in the design of plastic parts. They are now beginning to see that it can also help reduce the post-development delay in bringing a product to market.

While rapid injection molding is not identical to traditional production tooling, it is similar enough in process and technology to solve several problems and help speed up production. First, in addition to proving the design itself, it confirms that a part can actually be molded. Second, while production-tool molding can incorporate capabilities that rapid injection molding can't—internal cooling lines or sophisticated venting, for example—adapting a part to the demands of rapid injection molding by equalizing wall thicknesses and maintaining draft can actually simplify and speed up the manufacture of production molds while reducing their cost. In other words, rapid injection molding doesn't just produce prototype parts; it prototypes the production method that will produce those parts, allowing avoidable problems to be eliminated before the start of final mold making.

Perhaps even more important in today's fast-moving markets, tools made for rapid injection molding can also be used to mold parts in actual production resins and in production volumes—thousand or even tens of thousands of parts—while the "official" production tools are being made. In other words, yesterday's prototyping molds can produce today's "go-to-market" parts while tomorrow's ultra-high-volume molds are being made.

In fact, once it becomes clear that you can take “prototype” parts to market, you may actually find reasons to simply postpone the production of steel tools. One reason might be the ability to reduce up-front expenditures by ordering parts in smaller quantities than you commit to when you turn to production molds. This makes particular sense if there is any uncertainty about market demand for your new product. It’s a way of going beyond mere market tests and actually releasing your product to the market and gauging response before committing to

full-scale production. If the market’s reaction to your product suggests the need for “tweaking,” you can make changes quickly and be back on the market in days with an improved product. If necessary you can repeat the process several times, each at modest cost, before committing to mass production. In a sense, this sort of bridge tooling lets you treat a physical product in much the same way that software developers treat theirs, with versions tumbling onto the market one behind another as features are added. There’s really no reason that releases of Widget

Mark I, Mark II and Mark III have to be years apart if the market really wants an improved product. For those used to traditional methods this may be a novel approach, but if it eliminates that painful wait while production tools are being made it may be worth a try. And if it saves you the cost of sending tens of thousands of parts along with costly molds to the landfill because the market wants something slightly different, so much the better.

Crush Ribs — Getting to the Point

Press fits for injection-molded parts can be challenging. A well-designed injection-molded part will usually have draft, but the same draft that helps eject a part from the mold may also keep a press-fit part from staying firmly engaged. Consider a gear pressed onto a shaft (see Figure 1). In this case, the shaft has a D-shaped profile that acts as a drive flat and an orientation feature for assembly.



Figure 1: “D”-shaped through-hole in center of gear part accepts the drive shaft.

The drive flat keeps the gear from spinning freely on the shaft under load. Your molder will likely ask for draft on the D-hole to help release the feature from the mold. Draft is needed since as the resin cools, it shrinks onto the core that creates the hole. The request for draft is reasonable, but what if your design intent does

not allow for draft? Here are a few options.

1. Leave the hole as-is and require the molder to support zero draft. This is a risky request. If the hole is shallow enough you might not have any problems, but as the hole gets deeper, more stress is applied to the core of the mold during cooling and ejection. Increased force required for ejection could result in “pin punch” on the part, and increased ejection forces could result in breaking the core or ejector pins. The molder may have to tweak process parameters to prevent damage to the mold. This can increase the likelihood of imperfections like sink, porosity, and weak knit lines. Considering all the possible consequences, it makes sense to consider other options instead.

2. Add draft to the hole. Draft allows the part to release from the mold because the shrinking part does not have to be forced along the same diameter shaft for the depth of the hole. If the hole is drafted, once the ejector system gives a slight bump to the part, the part releases from the mold because the draft falls away from the part wall. This reduces stress on both the part and the mold. Having draft allows the molder the flexibility to use process tweaks to effectively address other geometric and cosmetic concerns. Draft solves the production problems, but your responsibility as the designer is to ensure that the draft doesn't adversely affect the function of your assembly. For example, you would not

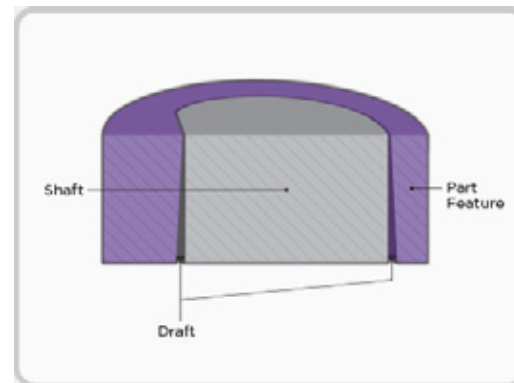


Figure 2: The cross section shows a portion of the assembly cut in half. Notice the draft on the purple plastic through-hole toward the bottom of the image. This could allow wobble in the finished assembly.

want to have slop in the fit that could cause gear wobble (see Figure 2).

3. A good compromise option is to add crush ribs. Crush ribs can give you the best of both approaches: draft for the molder and the same alignment you'd get with a straight-sided hole. The main hole is drafted for ease of ejection and protection of the mold, and three or more undrafted ribs along the length of the hole create a tight fit and good alignment with the shaft (see Figure 3).

Continued on next page...

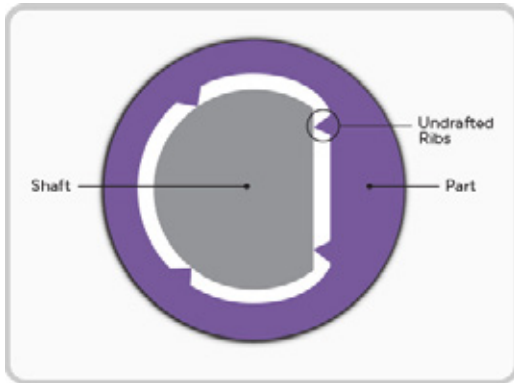


Figure 3: Traditional Crush Ribs. The sharp points of the plastic are undrafted ribs, while the main internal diameter of the hole has draft.

Because the ribs present only a small surface area to the mold, their lack of draft creates little resistance to ejection and less risk of damage to the mold. The narrow points where the ribs meet the shaft (or other mating part) allow the ribs to deform during press-fit to ensure a tight fit without creating a lot of stress on the part. Unfortunately, the sharp “V” features cannot be directly milled into the mold and thus require EDM or other extra processing during mold-making.

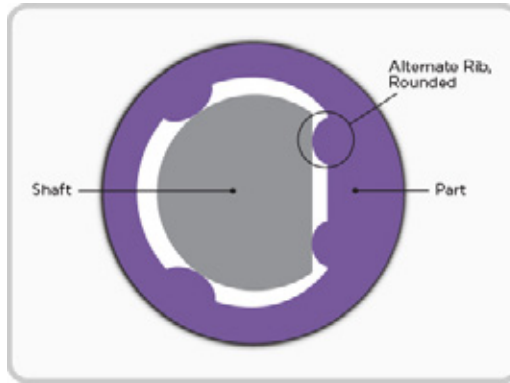


Figure 4: Suggested Crush Rib Design. The contact points of this design are formed by the radius of the end mill, allowing the inside diameter to be drafted while the ribs remain undrafted.

All is not lost, however. Protomold suggests an alternative rib shape that can be directly machined into the mold, minimizing cost. In place of the sharp point of the traditional V-shaped rib, consider a rounded rib (see Figure 4). A rounded rib can be directly created by 3-axis milling, eliminating the need for EDM electrodes. It is faster and less expensive than a V-shaped alternative, and still provides the narrow, “crushable” point of contact with the mating part.

The same form can be used for ribs designed to create “standoff” where parts mate, for example where an air gap is needed. While Protomold offers EDM in some applications, we believe the full-radius rib created by an end mill is just as effective and reduces cost by minimizing mold complexity and manufacturing time. This reduction in time allows you to receive parts faster, test sooner, and bring your ideas to market before your competition.