Rapid Injection Molding JOURNAL™

summer 2004

In this issue:

Breaking the Mold

Change is rarely a uniform process. That's particularly true in business, where entire industries are built around existing processes and technologies. When something new comes along, it takes a while to determine where and how it fits, who it will benefit, and what adaptations will be required. Then, sometimes, everything changes.

Examples are everywhere. Cars were once handmade toys for the rich. Cell phones filled entire briefcases. And the computing power you'd find today in a ten year-old's backpack once cost millions of dollars.

What we've created here at Protomold may not be the car, the telephone, or the computer, but by radically altering the economics of injection molding we are making a real difference in our customers' businesses. Those customers are in fields ranging from medicine to electronics to aerospace. And we're saving them both time and money, neither of which is a small matter in today's fast-moving, competitive markets.

We believe that rapid injection molding has the potential to change industries. We've been working with customers to help them determine how our technology can best serve them while, at the same time, continuing to develop and refine our processes to better fit their needs. The purpose of this journal is to expand that process, and we welcome your feedback on anything you see here.

Brad Cleveland
President & CEO

bradc@protomold.com

Fast, Cheap or Good: Choose Any Three

Most of us know the original version of this old cliché – the one that says "Choose Any Two." That reality – that you can't have everything, that you get what you pay for – is well known to product developers. As a gentle reminder it is more relevant to those who haven't been in the trenches and sometimes forget that finite resources must be allocated.

But what if you could have it all? It would certainly simplify a developer's life. It could shortcut the entire development process, be extended to production, and speed products to market. It would impact budgets – maybe even your bottom line – and in today's competitive markets, even a small edge can make a big difference.

But the question remains: Is "Choose Any Two" a law of nature or are there exceptions?

The answer is that there are exceptions, and we've all experienced them. As CAD replaced manual drafting, drawings became fast, cheap, and good. (Of course, expectations eventually caught up with the technology, so that now instead of being asked why we can't produce drawings in days, designers are asked why they can't be done in hours.)

As new technology emerges, however, there is always a time during which early adopters can choose fast, cheap, and good. Rapid injection molding (RIM) is just such an advance. Until now, you could wait weeks for conventional injection molded parts, pay exorbitant tooling charges, or turn to rapid prototyping, which produces parts that are typically less than functional and expensive in any significant quantity. RIM, on the other hand, produces fully functional injection molded parts, does it in days rather than weeks, and typically runs one-third the up-front cost of conventional injection molding.

In short, you can have it all. But leave up the "Choose Any Two" poster. It never hurts to remember that you are performing miracles.

What exactly is

RAPID INJECTION MOLDING?

Rapid Injection Molding typically cuts the lead time for the initial parts to one third of conventional methods.

Rapid injection molding, or RIM, is a lot like conventional injection molding (CIM), except that the design and manufacturing steps of mold-making have been automated. Because these two steps are the most costly and time- consuming in conventional molding, their automation is what makes RIM "rapid" and the reason it is so cost effective.

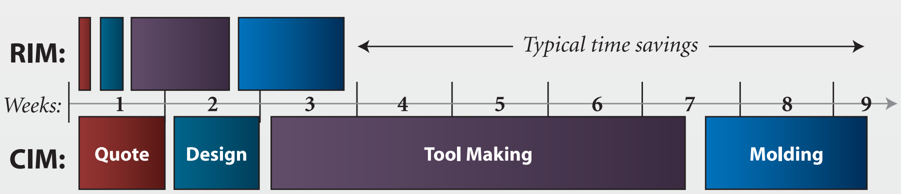
RIM typically cuts the lead time for the initial parts to one-third of conventional methods. Cost saving varies with the number of parts being produced, but RIM typically has a substantial cost advantage in runs of up to thousands of parts.

What do you give up? Actually, very little. RIM produces quality molds using advanced aluminum alloys and precise, high-speed CNC machining. Parts can be molded in almost any engineering grade resin.

What's the catch? None, as long as the mold for your part can be cut using three-axis CNC milling. Ideally your part should be designed without any undercuts, but the good news is that the design of many parts which appear to require them can be easily "tweaked" to eliminate the need for undercuts. The even-better news is that Protomold's online quoting software can help identify changes that will allow your part to be produced using fast, cost-effective RIM. ProtoQuotes® including moldability analysis are free; send your design and see for yourself.

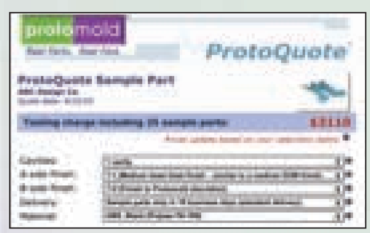
PROCESS SPEED :

Rapid (RIM) vs. Conventional Injection Molding (CIM)



Step by Step

1. Create a 3-D design.
2. Upload to Protomold at www.protomold.com.
3. Receive a ProtoQuote®
   (See sample at www.protomold.com)



4. Review moldability analysis and, if necessary, submit revised design for quote.
5. Select options – delivery, material, finish, etc. – and place online order.
6. Confirm order by phone with PO.
7. Review and approve gate and ejector pin layout.
8. Receive finished parts.

CASE IN POINT

Tuning up Product Development at Harmonic

Harmonic Inc., a provider of broadband optical networking, digital video, and IP delivery systems, shaved months off its product development cycle and saved thousands of dollars using rapid-injection molding rather than conventional methods. For its high-end transmitter, Harmonic wanted a high-tech look to match the product's advanced technology, along with the ability to re-use the look in related products. The appearance of the front bezel for the company's flagship transmitter system was extremely important, but as a primarily decorative element, it needed to be low in cost.

Harmonic originally considered a sheet-metal bezel, but tooling costs for the product's complex curves and cutouts were prohibitive. A second option was conventional injection molding with in-mold decorating, but this was problematic as well. Then, Harmonic discovered Protomold and rapid-injection molding. Intrigued by the fast turnaround, low tooling cost, and production-quality parts, Harmonic decided to test Protomold's capabilities with the production of a tray for managing fiber-optic cabling.

Harmonic uploaded a 3-D CAD model through the Protomold Web site, got a quote back within hours, and had finished parts in 15 days. "With Protomold's advanced software automating price-quotes and the manufacturing of molds, we avoided the costly, time-consuming custom engineering tasks that normally go into the development of injection-mold tooling," said Henry Baum, manager of mechanical engineering at Harmonic.

For Harmonic, the timing was perfect. Company engineers working on other components were also finding that in-mold decorating would not produce the accuracy and consistency they needed. In addition, it would take 12 to 16 weeks to cut the tools and produce the first pieces at tooling costs of $25,000 to $50,000.

The company again turned to Protomold. Within hours of submitting a 3-D CAD model of its bezel, Harmonic received an interactive ProtoQuote® including estimated costs and illustrated suggestions for improving moldability. As before, Harmonic received parts in 15 days and saved thousands on tooling. But more importantly, the company was able to make up time lost pursuing other options and get its schedule back on track. "Protomold saved us about 16 weeks of development time and $40,000," says Baum.

Rapid-injection molding simplified design improvement as well. For example, when Harmonic displayed equipment with the rapid-injection molded bezels at a trade show, feedback suggested the need for design enhancements. Recutting a conventional injection-molded tool would have been cost-prohibitive. But with Protomold, creating a new tool was quick and cost-effective. Harmonic had discovered a key benefit of rapid injection molding: the technology makes it possible to go from a 3-D model to real, injection-molded parts in a single step. By cutting costs and lead times and allowing companies to test real parts, it frees design engineers to focus on innovation and quality improvement.

"With Protomold's advanced software automating price-quotes and the manufacturing of molds, we avoided the costly, time-consuming custom engineering tasks that normally go into the development of injection-mold tooling."

TIPS for better PROTOTYPING

BUYING PROTOTYPE PLASTIC PARTS CAN BE A CHALLENGING EXPERIENCE. HERE ARE SOME TIPS:

1. Understand your technical requirements.

Start by defining form, fit, and function. Form includes size, desired level of cosmetics and, if necessary, color. Fit consists primarily of the required dimensional accuracy. Functional requirements include operating temperature and strength properties such as tensile strength and hardness. Finally, determine, as nearly as possible, the number of pieces you will need.

2. Know your business constraints.

If you need just a few prototypes overnight, you can choose from a variety of rapid prototyping technologies. But if you need fully functional prototype parts and your turnaround time is days or a week, rapid injection molding (RIM) is probably your best option.

3. Understand the strengths of various prototyping methods.

Today's options include rapid prototyping, rapid tooling, and rapid injection molding. Rapid prototyping (RP) produces complex shapes quickly. The materials differ from those in production parts and, because cost does not decrease with quantity, it is typically used for 10 or fewer parts when lead-time is five days or less. Rapid tooling (RT) typically takes five to 10 days, produces parts that are still not functionally identical to production parts, and can be cost effective for up to 100 pieces. If functional testing is planned, rapid injection molding (RIM) can cost-effectively produce hundreds or even thousands of production-quality parts in five to 15 days.

4. Make sure your prototyping method can accommodate your design.

Napkin sketches or 2-D drawings don't provide the data needed by today's online quoting and advanced prototype-manufacturing processes. Use a 3-D CAD format instead. Common 3-D file formats include STL (stereolithography), IGES (initial graphics exchange specification), and STEP (standard for the exchange of product). Most CAD programs can output these formats. If all you have is 2-D capability, some RP service bureaus can convert 2-D to 3-D CAD format for prototyping.

5. Watch out for middlemen.

Service bureaus manufacture prototype parts in-house from 3-D models and ship them directly to you. Typically, this arrangement will give you direct access to the people responsible for the production of your parts. Brokers, on the other hand, typically do not own equipment and will outsource your order to someone else. This may work fine, as long as your design is perfect, the schedule is fixed and no unexpected technical challenges come up (like that ever happens).

6. Select the best-qualified prototyping service provider.

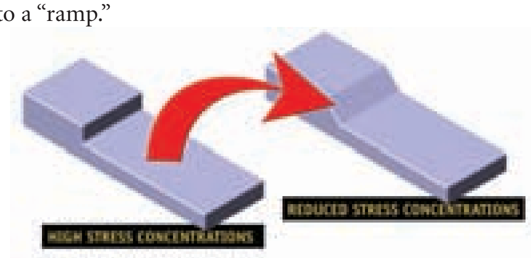
Look for a vendor who will, if necessary, help you learn the ropes, and assist with decisions regarding technology. Along with basic requirements, consider a service provider's ability to handle unexpected technical problems. Get references and ask about the provider's service and technical expertise. Keep in mind that automated online quoting can speed the quoting process but doesn't replace person-to-person customer service.

DESIGNER'S CORNER

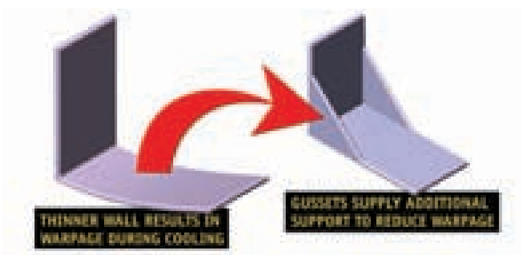
Planning for Moldability

The secrets to designing an injection-moldable plastic part are all in the details — some of which are described below.

Molten plastic shrinks as it hardens. This can lead to "sink" – dips in surfaces – and warping where areas of different thickness meet. You can reduce "sink" by avoiding unnecessary thickness in part walls. Warping at "stair steps" can be reduced by turning the stair step into a "ramp."



Warping of flat walls can be controlled by adding reinforcing gussets at the corners or ribs along their length.



"Ribs" are an effective way to stiffen walls, but the rib itself can cause sink in the face of the wall it reinforces. You can prevent this by making the rib approximately half (40-60%) as wide as the thickness of the wall it reinforces.

Sharp corners create "stress risers" – points of internal stress at which part failure can occur. Rounding these corners eliminates the stress. The inside of curves should have a radius (r) of at least one-half the thickness (T) of the wall in which they occur (r= 0.5*T). Part walls parallel to the direction in which the part is pulled from the mold can make removal difficult. "Vertical" walls should therefore be drafted, i.e., tapered to pull away from the mold wall as the part is removed.

Draft Guidelines

- **Minimum draft for vertical walls : 0.5 degrees**
- **Preferred draft for smooth surfaces : 2 degrees**
- **Minimum draft for light textured surfaces (PM-T1) : 3 degrees**
- **Minimum draft for heavily textured surfaces (PM-T2) :5 degrees**

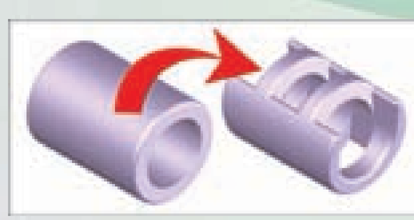
Also, to prevent damage to both molds and parts, the minimum draft of metal-on-metal sliding surfaces should be at least three degrees.

Rapid Injection Molding most easily accommodates "straight pull" molds. These are two-part molds that open by pulling mold sections in opposite directions. For this reason, optimal RIM part design does not include "undercuts" (features in which plastic would be trapped behind the metal of a straight pull mold as the mold is opened). Fortunately, many undercuts can be avoided by redesigning or simply realigning the part. (See sidebar, Making it Fit)

For more information on designing parts for both conventional and rapid injection molding, visit www.protomold.com/designguide.

Making it Fit

A "through-hole" like the one in the accompanying diagram cannot be formed in the straight pull molds used in RIM.



However, redesigning the fully enclosed hole into one that is open on alternating surfaces allows the part to be easily removed from a two-part mold. Alternatively, the part can be molded without the hole and the hole drilled as a separate operation.

In this issue of the Rapid Injection Molding Journal we have incorporated an excellent article originally published in the April 2004 issue of Injection Molding Magazine. We hope you also find it interesting and useful.

Designing *with* ABS

ABS IS A GOOD MATERIAL FOR APPLICATIONS WHERE LOWER-COST COMMODITY PLASTICS DO NOT PROVIDE THE REQUIRED COMBINATION OF IMPACT STRENGTH AND STIFFNESS.

In this bimonthly column, Glenn Beall of Glenn Beall Plastics Ltd. (Libertyville, IL) shares his special perspective on issues important to design engineers and the molding industry.

If you used your telephone today you touched an injection molded ABS part.

The excess polystyrene capacity available following World War II resulted in modifications of that material to capture a broader market. A 1948 patent was granted to U.S. Rubber Co.'s L.E. Daley for an ABS type of material. These first ABS plastics were mechanical blends. They found their largest market in extruded pipe and molded fittings. These were not the processor-friendly materials of today.

Continuing research in the 1950s allowed the styrene-acrylonitrile copolymer to be grafted onto the crosslinked polybutadiene. With that breakthrough ABS became acceptable for many applications. Production of ABS was 65 million lb in 1960, 550 million lb in 1970, 1 billion lb in 1980, 1.161 billion lb in 1990, 1.451 billion lb in 2000, and 1.315 billion lb in 2002.

ABS is a terpolymer identified as acrylonitrile-butadiene-styrene. This is a mouthful, so the material is universally referred to as ABS. Acrylonitrile provides resistance to heat, chemicals, and aging. Butadiene accounts for ABS's toughness and low-temperature impact strength. Styrene contributes stiffness, surface luster, ease of processing, and low cost. Changing the type and amount of these three ingredients allows ABS to be adapted for a variety of end-use requirements.

ABS was originally created as a bridge material to fill the gap between low-cost commodity materials such as PE and PS, and higher-cost PC, nylon, and acetal engineering plastics. The list price for large quantities of ABS is $.62 to $1.05/lb for special grades, or $.024 to $.039/cu in. ABS is an amorphous thermoplastic known for its rigidity, dimensional stability, lustrous abrasion-resistant surface, color consistency, ease of processing, and midrange cost. Its defining characteristic is its notched Izod impact strength of 2.3 to 12 ft-lb/in, coupled with its good balance of other properties. Most ABS compounds are opaque, but there are a few transparent grades with light transmission of 72% to 80%.

APPLICATIONS
It is now possible to vacuum metallize or electroplate most materials. ABS was, however, the first plastic that formed a secure bond to electroplated coatings. This led to two large markets: appearance-type plumbing applications such as water faucets, and automotive trim parts including grilles and wheel covers. Plumbing and automotive are still major markets, but the interior trim business has been replaced by polypropylene thanks to carmakers' efforts to reduce cost (and quality?). The large pipe and fitting market has been eroded by lower-cost PVC.

Other major markets include office furniture and machine housings; household furniture; appliance and electronics housings; floor care products; refrigerator liners; luggage; toys; and recreational products such as camper bodies, boat hulls, and snowmobile shrouds. Medical products represent a significant market. Large quantities of ABS are extruded into sheet for thermoforming.

Reprinted from Injection Molding, April 2004 • Copyright © 2004 Canon Communications LLC

SPECIFICS OF ABS DESIGN

Wall thicknesses *of .010 inch have been molded, but ABS is not an ideal material for thin-walling. A better minimum thickness is in the range of .030 to .040 inch. Part thicknesses greater than 1.000 inch have been successfully molded using a warm mold with large gates and runners. The ideal compromise between cost and quality will be realized with a maximum wall thickness of .250 inch.*

Radiusing*. ABS is not a notch-sensitive material; however, radiusing the corner of an injection molded part improves its strength by distributing corner stresses over a broader area. Inside corner radiuses should be limited to not less than 25% of the part's wall thickness. For maximum strength the radius should be 60% of wall thickness. Larger radiuses can be specified, but this will not significantly increase part strength.*

Molding draft angles. *ABS is a rigid, but relatively soft, material that can often be molded with no draft angle. However, a .5° to 1° draft angle per side will shorten cycle time and make a better part.*

Projections of all types can be molded with ABS. *The thickness of projections can be 75% of wall thickness. In instances where sink marks cannot be tolerated, the thickness of projections should be reduced to 50% to 60% of wall thickness.*

Depressions and holes. *Strong weldlines that do not distract from a part's appearance can be produced with proper molding conditions. Inside corners on irregularly shaped holes must be radiused to avoid molded-in stress. Draft angles on holes will improve quality and shorten cycle time. The depth of holes should be limited to two and one-half to three times the thickness of the core to avoid core pin deflection.*

Tolerances. *ABS is a low-shrinkage, dimensionally stable, amorphous material with uniform shrinkage. A "commercial" tolerance for a .125-inch-thick, 1.000-inch-long ABS part would be ±.0045 inch. A more costly "fine" tolerance could be ±.0023 inch. Parts that are less than .125 inch thick can be held to a slightly closer tolerance. Thicker parts, which take longer to cool and shrink more, may require a larger tolerance.*

WHAT'S NEW

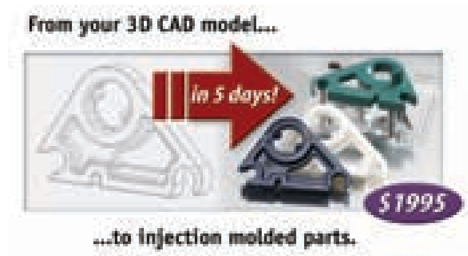
Protomold News

We like our new sign so much, we wanted to share it with you. It's even better at night, when it's backlit in Protomold green and purple. So the next time you're out driving through Maple Plain, Minnesota, look for us.

When we say "one-day turn-around" on ProtoQuotes®, we're not exaggerating. In fact, we're being a bit modest. Our newly expanded sales & customer service group is now averaging less than four hours after receipt of a 3-D CAD model to turn around a customer's ProtoQuote®.

And, speaking of statistics, our manufacturing group is processing 8-10 jobs per day, including the manufacture of about 200 molds per month, which is why we've had to expand our mold storage facility.

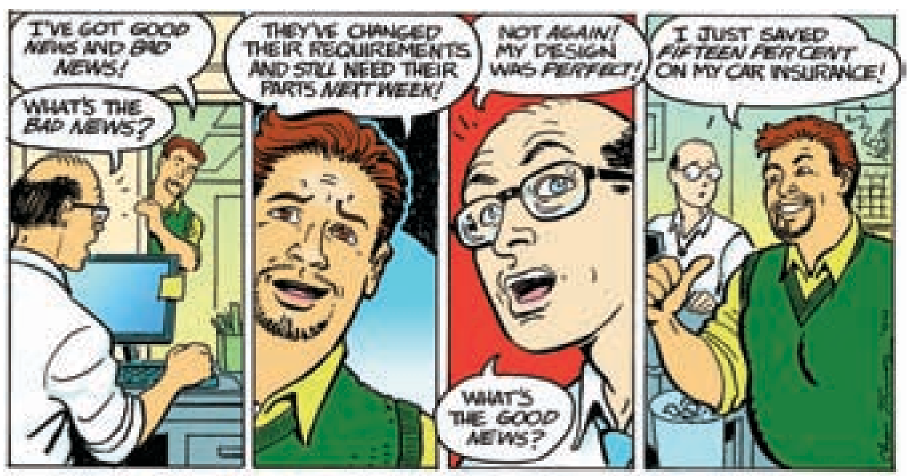
At Protomold, we bring together Information Age technologies and advanced mold-making to slash tooling and production times for prototype and low-volume plastic parts.



That's why we can mold your parts in the engineering resin of your choice faster and more cost-effectively than anyone else. Visit our web site to see application examples or request an online ProtoQuote®.

Everything by Design

Bringing new products to market can be a very frustrating experience, so it's a good idea to find humor wherever you can. Please email your suggestions for future "Everything by Design" cartoons to: marketing@protomold.com.



Rapid Injection Molding

protomold®

Real Parts. Real Fast. Real Savings!

www.protomold.com/parts

The Protomold Company, Inc.
1757 Halgren Road
Maple Plain, MN 55359

(763)479-3680