Digital eBook
A Design World Reference

Injection Molding with Protolabs

Design World

PROTOLABS®
Manufacturing. Accelerated.™

TechTIPS

Designing for Moldability: Fundamental Elements

A quick reference guide to wall thicknesses, surface finishes, materials, and other thermoplastic injection molding insights

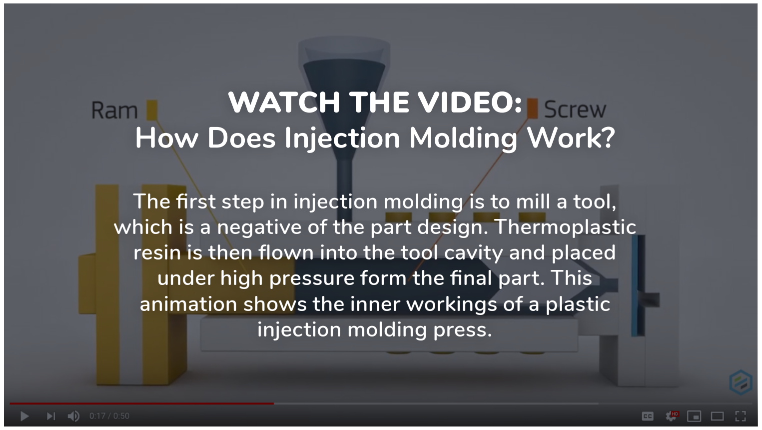
DESIGNING PLASTIC PARTS for moldability has always been important for traditional injection molding processes, but it's particularly beneficial during rapid injection molding to ensure speed and quality remain constant during manufacturing. This guide examines many of the important design considerations that are encountered during injection molding part design and product.

What is Rapid Injection Molding?

Rapid injection molding a technology-driven process that leverages manufacturing automation. CAD models are sent directly to the production floor where mold milling begins, but in most cases, molds are fabricated from aluminum, not steel. This allows for faster and most cost-effective tooling when compared to traditional steel molds.

There are number of surface finish options available and capabilities include side-action and hand-load inserts as well as simple overmolding and insert molding. Selective use of electrical discharge machining (EDM) is applied to improve mold features like corners and edges. The result is part production within a few weeks versus months when compared to traditional injection molding methods.

Here some applications that work well for rapid injection molding and some industries taking advantage of the technology:



Applications

- Iterate quickly with quick-turn prototypes
- Test functionality during product development with production-grade parts
- Test multiple materials
- Test multiple CAD models
- Make quick iterations
- Implement bridge tooling
- Leverage low-volume production for on-demand parts
- Manage demand volatility
- Get thousands of parts within days

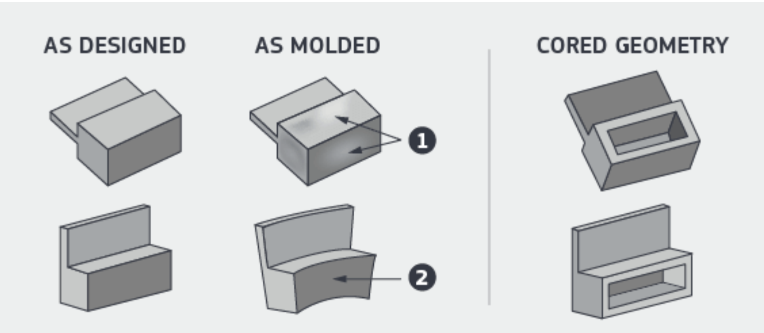
Industries

- Medical and health care
- Automotive
- Electronics
- Aerospace
- Unmanned aerial vehicles (UAVs)
- Consumer products
- Appliance
- Lighting
- Marine
- Robotics

From wall thickness and radii to ramps and ribs, let's look at all of the factors designers and engineers should consider if their parts will be injection molded.

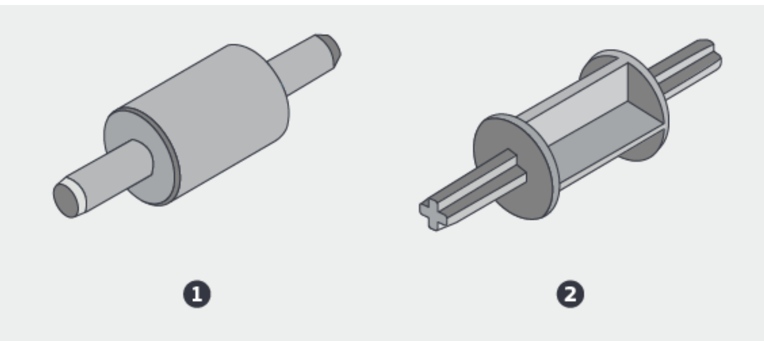
Wall Thickness

The most important design requirement for getting good molded parts: maintain constant wall thickness. Good injection-molded part design relies on consistent wall thickness to minimize the potential for warped or distorted parts.



Core Geometry

Core out parts to eliminate thick walls. You get the same functionality in a good molded part. Unnecessary thickness can alter part dimensions, reduce strength, and necessitate post-process machining.



Size – Maximum Dimensions (in inches)

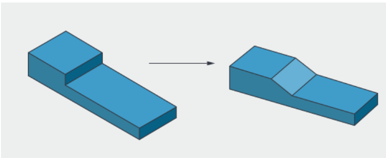
SIZE	18.9 in. x 29.6 in. x 8 in.
VOLUME	59 cu. in.
DEPTH	4 in. from parting line
	Up to 8 in. if parting line can pass through the middle of the part
PROJECTED MOLD AREA	175 sq. in.

Wall Thickness Guidelines (in inches)

MATERIAL	RECCOMENDED WALL THICKNESS
ABS	0.045 in. - 0.140 in.
Acetal	0.030 in. - 0.120 in.
Acrylic	0.025 in. - 0.500 in.
Liquid Crystal Polymer	0.030 in. - 0.120 in.
Long-Fiber Reinforced Plastics	0.075 in. - 1.000 in.
Nylon	0.030 in. - 0.115 in.
Polycarbonate	0.040 in. - 0.150 in.
Polyester	0.025 in. - 0.125 in.
Polyethylene	0.030 in. - 0.200 in.
Polyphenylene Sulfide	0.020 in. - 0.180 in.
Polypropylene	0.035 in. - 0.150 in.
Polystyrene	0.035 in. - 0.150 in.
Polyurethane	0.080 in. - 0.750 in.

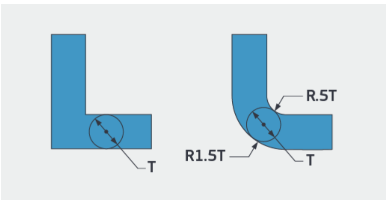
Note: These are general guidelines, subject to part geometry and molded construction. Larger parts shouldn't be designed with the minimum wall thickness. Protolabs' general rule for wall thickness is 0.040 to 0.140 inches. To see the chart in millimeters click here.

Designing for Moldability: Fundamental Elements | continued



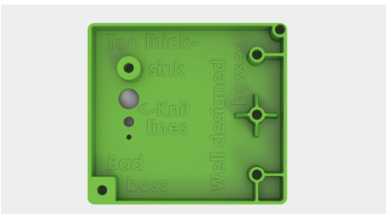
Ramps

Eliminate sharp transitions that cause molded-in stress.



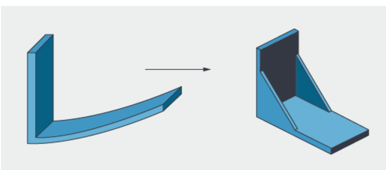
Radii

Sharp corners weaken parts, causing molded-in stress from resin flow. Adding radii in sharp corners is suggested.



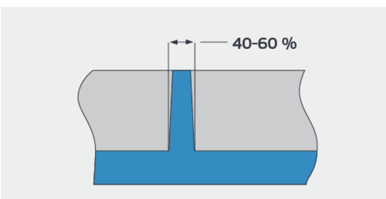
Bosses

Don't create thick sections with screw bosses since tick sections can cause sink and voids in your part.



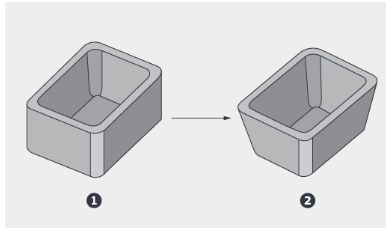
Fillets

Design part features that support themselves.



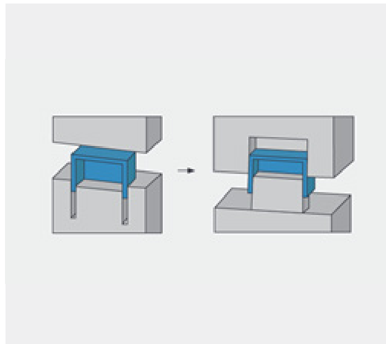
Ribs

To prevent sink, ribs should be no more than 60% of the wall's thickness.



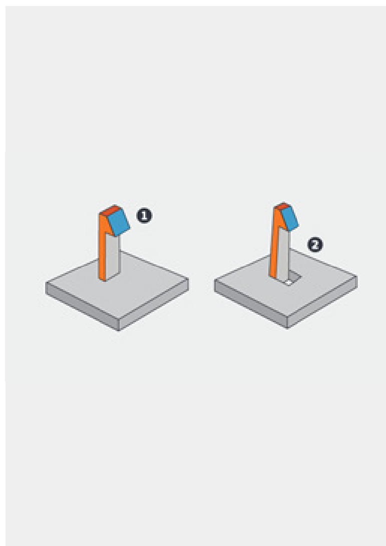
Draft

Draft (slope the vertical walls) as much as possible—this makes it easier to eject parts without drag marks or ejector punch marks.



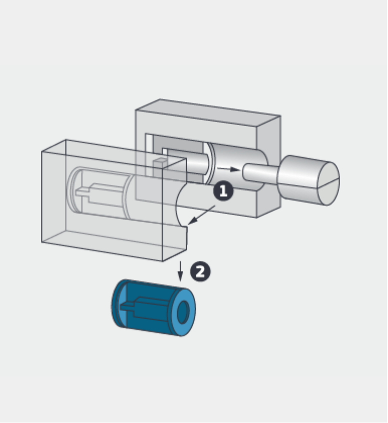
Core-cavity

When you draft, use core-cavity instead of ribs if you can. It allows you to have constant wall thickness rather than walls with a thick base. We can mill molds with better surface finish and deliver better parts faster.



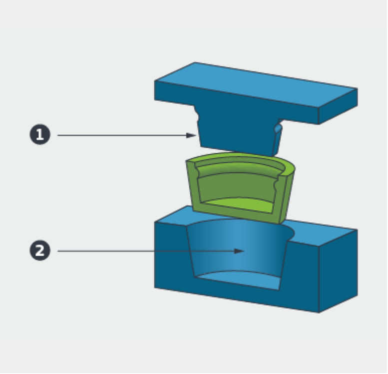
Undercuts

An undercut is an area of the part that shadows another area of the part, creating an interlock between the part and one or both of the mold halves. The left image (1) illustrates a clip with undercut feature. On the right image (2), an access hole beneath the undercut allows the mold to protrude through the part and provide the needed latch shutoff geometry.



Side-Actions

Side-actions can form undercuts on the outside of your part. Undercuts must be on or connected to the parting line. They must be in the plane of the parting line. Undercuts must be connected and perpendicular to the direction the mold is opening.



Bumpoffs

A bumpoff is a small undercut in a part design that can be safely removed from a straight-pull mold without the use of side-actions. Bumpoffs can be used to solve some simple slight undercuts, but are sensitive to geometry, material type.

Deep Milling

Draft the part as much as possible. This allows us to make deeper features for you. Draft allows us to reduce tool chatter and cosmetic defects when milling deep walls. If you can fit it in, use 1 degree of draft or more. On core-cavity designs, try to use 2 degrees or more. A rough rule of thumb is 1 degree of draft for each of the first 2 inches of depth. From 2 to 4 inches of depth, either 3 degrees of draft or a minimum of 1/8 in. thickness may be required.

Pickouts

A pickout is a separate piece of metal that is inserted into the mold to create an undercut. It is ejected with the part, then removed by the operator and re-inserted in the mold. Using a pickout overcomes many shape and positioning restrictions, but is more costly than sliding shutoffs, or using a side-action.

Steel Core Pins

These holes can be made with steel core pins in the mold. 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. 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.

WANT MORE?

CLICK HERE FOR THE FULL ARTICLE

Thermoplastic Material Selection for Injection Molding

Learn about material properties to quantitatively analyze parts before selecting a thermoplastic resin

THIS WHITE PAPER is aimed at an engineer who plans to quantitatively analyze a part, determine loads, stresses, strains, and environments and make an optimal material decision based on the analysis. If life safety is involved, or reliably or efficacy are absolutely required, every part should be engineered and materials selected accordingly. If you look through this paper and see the many factors involved and how environment and application influences material selection, you can understand why an engineer will be very reluctant to recommend a specific material for someone else's part.

However, many of Protolabs' customers who design parts are not engineers, and many applications of Protolabs manufactured parts are quite benign and are expected to stay well within the performance envelope of common plastics. If your application lives at room temperature, it doesn't have appreciable loads, and you're willing to make a few parts and whack them with a hammer to see if they're strong enough for your use, look for the simplified suggestions for selecting materials at the bottom, called "Don't Make Me Do the Math."

Material selection can be a guessing game. First, there is a general gap in understanding the fundamental relationship between the internal structure of the material and its properties. Second, accurately defining application requirements is usually given insufficient time and attention. Finally, even when these first two hurdles are overcome it can be hard to find accurate property data for materials.

The Standard Material Data Sheet

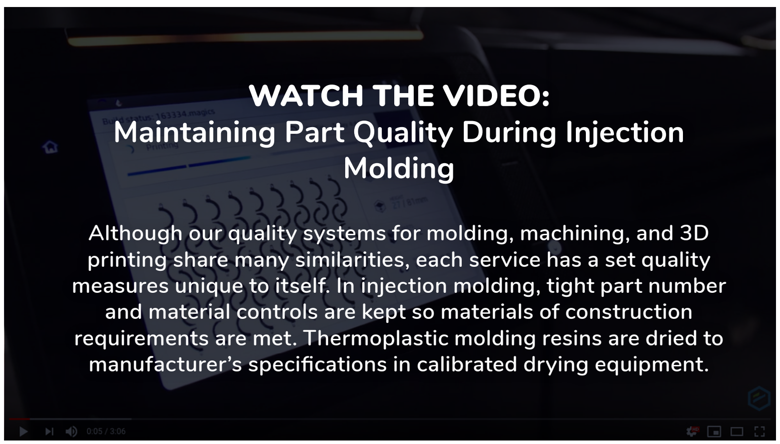
The standard material data sheet consists almost entirely of performance characteristics measured at room temperature. In addition, the performance characteristics are associated with catastrophic events that are not considered to be an acceptable outcome for engineered plastic products. Tensile strength at yield and elongation at break represent the standard metrics of material performance, but yield and break are not the desired responses of plastic parts when they are placed under load.

Determining the appropriate material for your application involves synthesizing information from a variety of incomplete sources.

The data-sheet is the primary source of information, and you should learn to extract as much information as possible from this source. Appendix A shows a data sheet for 30% glass fiber-reinforced PBT polyester. This is a good example of a reasonably detailed data sheet.

More than 85,000 commercial options for plastic materials are listed in materials databases, and the real number is probably over 90,000. This extensive set of options can be sorted into approximately 45 polymer families or blends, and these 45 families can be further divided into two broad categories: thermosets and thermoplastics. While thermosets were the first commercial polymers, their use has diminished to the point where they constitute only about 15% of all the material processed in a given year. Therefore, this paper focuses on thermoplastics.

More detailed information can sometimes be obtained from design manuals and application notes published by individual material suppliers and can fill in the gaps in the data sheet. Supplemental information is usually more available for higher performance engineering and specialty materials than it is for commodity materials. If you really want to understand a material you need to be prepared to do a little detective work.



Understanding the Maximum Short-Use Temperature

Maximum short-term use temperature is possibly the most important data sheet parameter. Traditionally, this is the deflection temperature under load (DTUL), also called the heat deflection temperature (HDT). Another related parameter is the Vicat softening temperature. Because DTUL measures mechanical deflection and the Vicat point is closer to the actual melting or softening point of the polymer, the Vicat number will typically be higher. For a material such as the glass reinforced PBT of Appendix A, which is a semi-crystalline material, all of these values will be very close to the crystalline melting point of the polymer, 223°C (435°F). Any application that involves even momentary excursions above this temperature will eliminate this polymer from consideration.

The upper-temperature limit for filled or unfilled amorphous polymers can also be found by looking at HDT or DTUL. For example, for unfilled polycarbonate the HDT values range between 130–140°C depending upon the grade. Vicat softening points, where provided, are a few degrees higher. Amorphous polymers do not show a significant crystalline structure when they solidify, therefore they do not have a melting point. However, they do exhibit something called "glass transition." From a practical standpoint, this is the temperature at which amorphous polymers lose their load-bearing properties.

For polycarbonate this value, when measured by dynamic mechanical methods, is approximately 153°C, just a few degrees above the Vicat softening point and 10–20°C above the DTUL, depending upon the specimen geometry and how the DTUL is measured. Vicat softening temperatures and DTUL values should never be used as long-term performance characteristics. However, they can be used to gauge short-term heat resistance when short-term is defined in minutes. Any application environment that involves excursions to temperatures above these properties will eliminate that particular material from consideration regardless of any other attributes it may possess.

Yield and Tensile Strength

Long-term performance when a material is under constant stress involves a property called “creep resistance;” if the stress is periodic then fatigue resistance becomes the dominant consideration. The relationship between stress, time, and temperature is complicated and frequently the data needed to make good decisions about long-term behavior of a material under load are not available. Here again the data sheet can provide an upper limit. The upper limit for ductile materials is the yield strength of the material and for brittle materials it is the stress at break. Both values define the point at which the material fails catastrophically. Any environment that involves stresses and strains higher than these values eliminates the material from even short-term consideration. Beyond this simple filter, you will next need to look at long-term temperature effects.

Understanding the Relationship between Stress and Temperature

Predicting long-term elevated temperature performance requires access to multiple data points. As the temperatures increase or as the desired lifetime of the product increases, the allowable stress level at which the material can be used decreases according to a function that is dependent upon the thermal and mechanical properties of the specific material. Correlations between short-term and long-term performance have shown that the long-term working stress levels for a thermoplastic will typically be between 20–40% of the short-term strength at yield or at break. Unfilled materials tend to fall in the lower end of this range while highly filled compounds tend to fall into the upper end of this range. Safety factors for a given product will reduce these values and if the temperature of the application environment approaches the DTUL values it is possible for the sustainable working stress to be only 3–5% of the value provided on the data sheet. Some data sheets will give tensile strength and modulus values at multiple temperatures. If available, these data can eliminate a lot of guesswork. Table 1 gives an example of tensile strength values provided at multiple temperatures for a glass fiber-reinforced nylon 6/6.

TABLE 1: TENSILE STRENGTH AND MODULUS OF 43% GLASS-FILLED NYLON 6/6 AT MULTIPLE TEMPERATURES

Temperature	Tensile Strength (psi)
-4°	36,500
23°	30,000
77°	17,500
121°	12,500

Understanding the Relationship between Temperature and Aging

All polymers have a long-term sensitivity to oxygen and this sensitivity increases at higher temperatures. Degradation associated with aging is captured by a property called the “relative thermal index,” or RTI. This value comes from a test mandated and administered by Underwriters Laboratories. It is currently the best gauge for measuring the long-term effects of aging on the mechanical and electrical properties of polymers. RTI testing begins by measuring key baseline properties such as tensile strength, notched Izod impact resistance, and arc resistance. Test specimens are then aged at multiple temperatures and the baseline properties are monitored until they decline to 50% of the original values. The time required to reach 50% performance is called the "time to failure." If three or four aging temperatures are used and the logarithm of the time to failure is plotted as a function of reciprocal temperature, the data points can be fitted to a straight line. This line is then extrapolated to a standard time (normally, about eight years) and the temperature predicted to cause failure at the standard time is the relative thermal index. For most thermoplastics the RTI values are lower than DTUL and Vicat softening values. This is the case for the glass-filled PBT in our Appendix A sample data sheet where the DTUL and Vicat values are all above 200°C (392°F) while the RTI values are 140°C (284°F). However, it is possible for soft, flexible materials with good oxidative stability, such as PTFE, to have RTI values that are higher than their DTUL numbers. RTI values can be used to predict long-term performance where aging is the primary concern.

The aging process follows an empirical rule that relates degradation to temperature. The rate of degradation doubles with each 10°C increase in temperature. This is an exponential relationship so a change of 20°C will increase the degradation rate by a factor of 2^2 or 4 while an increase of 30°C increases the rate by a factor of 2^3 or 8. Since RTI is indexed to a time frame of approximately eight years, you can estimate that a material could survive for four years at a temperature 10°C above the RTI, two years at a point 20°C above the RTI, and one year at a point 30°C above the RTI. Safety factors should be built into this calculation since studies show that the actual acceleration factor, while nominally 2, can vary from as low as 1.8 to as high as 2.5.

Modulus

Modulus is provided on virtually every data sheet. Most often this is provided as tensile modulus or flexural modulus. The modulus relates stress to strain and can be thought of as a measure of stiffness. In most cases the modulus is calculated in the linear region of the stress-strain curve. Linearity often is lost at very low strains. Figure 1 shows a magnified view of the early portion of a stress-strain curve for a highly-glass-fiber-reinforced nylon 6/6. While the modulus of this material at room temperature is given as 10600 MPa (1,537,000 psi), the graph shows that the stress-strain plot departs from linearity at approximately 0.4%. Beyond this point each incremental increase in stress produces a progressively larger corresponding strain. Figure 2 shows that while the slope of the modulus line reflects the value provided on the data sheet, the effective slope of the line connecting the origin to the yield point has a slope that is only 40% of this reported value. Therefore, when using the modulus as a selection property it is important to understand the position of the application stress on the stress-strain curve. As application stresses approach the yield point the expected lifetime of the product declines. Table 2 shows the maximum operating stress for a polycarbonate material as a function of time at two different temperatures. At very short times, less than an hour, the stress limit is nearly the same as the yield stress for the given temperature. As the time frame of the application increases under load, the maximum allowable working stress declines.

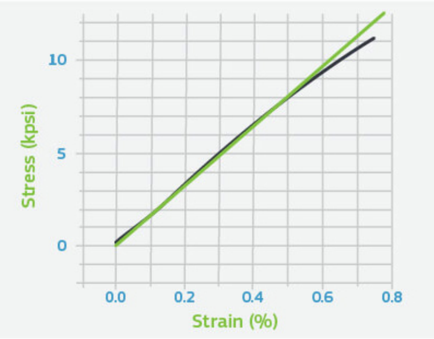


Figure 1: Initial portion of tensile stress-strain curve for 43% glass-fiber reinforced Nylon 6/6.

Cosmetic Defects in Injection Molding and How to Avoid Them

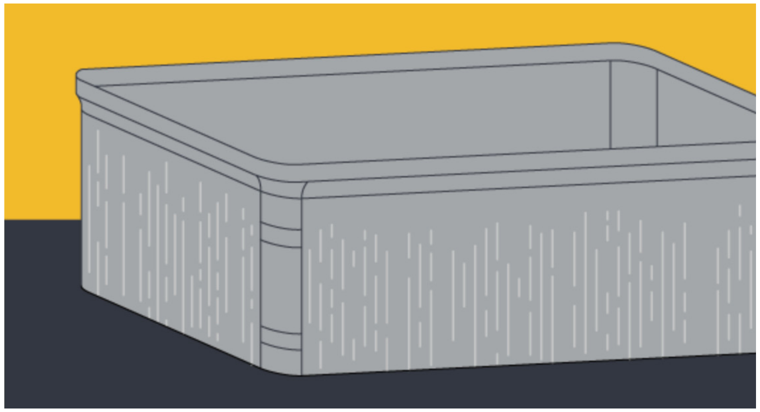
Improve the appearance and performance of injection-molded parts by eliminating cosmetic issues early

AS WITH ANY manufacturing process, injection molding comes with its own set of design guidelines, and design engineers who understand these best practices will increase their chances of developing structurally sound and cosmetically appealing parts and products.

Here are some common cosmetic defects that occur on plastic injection-molded parts, and tips on how to avoid them.

Avoid Sink in Molded Parts

As its name implies, sink appears as a dimple or shallow depression on the surface of a molded part. It's caused by thicker than normal cross sections, non-uniform part design or an improper gate placement—the doorway through which hot plastic first enters the mold cavity. Some plastics—polypropylene and acetal, for example—are very susceptible to sink, whereas fiber and glass-filled materials are less prone to sink. At Protolabs, we have wall thickness recommendations for each material, and advise that a workpiece minimum wall thickness be no less than 40 to 60 percent of its thickest section. Material flow within the mold should travel from thick to thin whenever possible, which might mean reorienting the mold cavity, or placing the gate in an area originally reserved for a cosmetic surface.



Drag or scrape lines may occur on a part if sufficient draft is not incorporated into the design of the vertical walls—those part surfaces parallel to the direction of the mold operation.

Elude Warp When Using Injection Molding

Design a part with walls too thin for the target material and it's likely to curl up like a potato chip. This is called warp, and is easily avoided by following the same rules used with sink, namely staying within the general wall thickness guidelines. Ironically, the glass-filled materials that work well with sink-prone parts are more susceptible to warp. That's because, as the part cools, the glass fibers tend to line up in the same direction, creating internal stresses. Parts with internal support structures—gussets to support thin walls, or ribbing of large flat surfaces—fare best against warp.

Avoid Drag in Molded Parts

Sufficient draft is an important part of any mold design, and quick-turn tooling is no exception. Vertical walls, meaning those part surfaces parallel to the direction of mold operation, should have a minimum draft angle of a 1/2 degree, and 2 degrees is even better; heavily textured surfaces may require 5 degrees or more. Without proper draft, part ejection becomes difficult if not impossible, and drag or scrape lines may occur.

Flash in Molded Parts

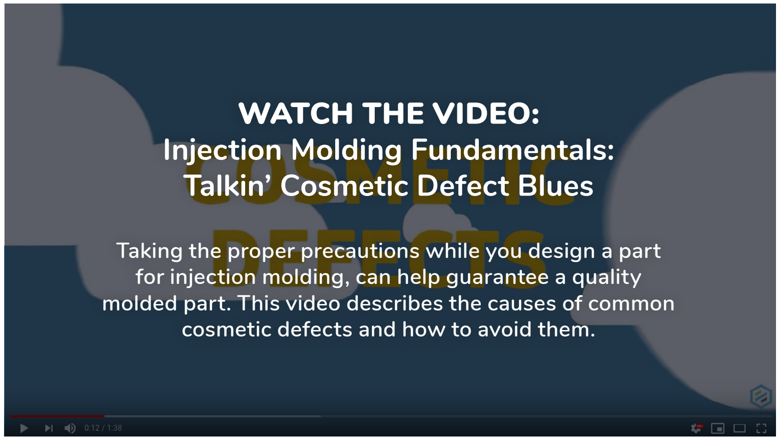
Look closely at a rubber O-ring and you'll see a thin line of material at its outermost periphery. That's a parting line, the seam where the two halves of the mold come together. With free flowing materials such as Santoprene or unfilled nylon, a small amount of flash can sometimes ooze into the seam, and often requires trimming once the part has cooled. On a donut shape such as this, there's little choice over the parting-line location, but many orthogonal parts have sharp corners, which make a clean, crisp junction at which the mold can separate. Flash or no, you should expect a parting line on most molded products, but we will identify the parting line location on your ProtoQuote, and may suggest ways of modifying the part geometry to avoid one.

Avoid Swirling in Injection Molding

From Honey Beige to Cornflower Blue, we stock more than 40 standard colorants. These are mixed with natural resin pellets immediately prior to the molding run and are usually quite close to the target color, but the final product may vary due to the polymer being used, texture and polish of the tool, and swirling during the mixing process. If you want an identical color match on your parts, it would be best to purchase color-matched, pre-compounded resin from an external vendor. We accept most customer-supplied resins sent our way.

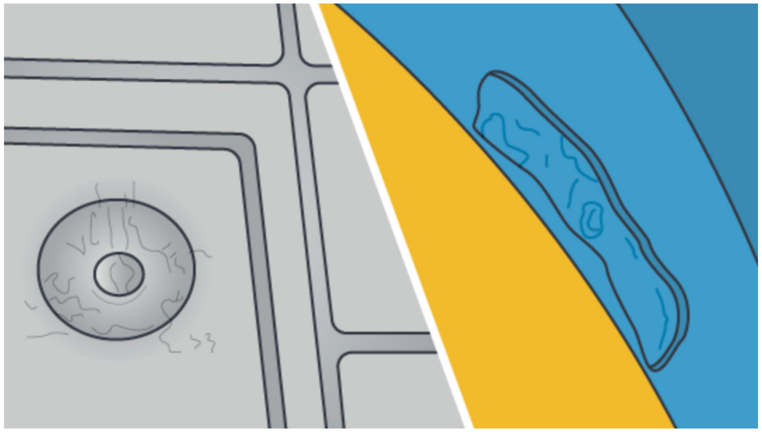
Knit Lines and How to Solve Them

Worried about those fine lines that look like hairline cracks in your injection-molded part? Don't be. Those are knit lines, formed when two opposing flows of material join together in the mold cavity. Commonly seen at the edge of a hole or other cored feature, knit lines are—as a rule—purely cosmetic, but may create a physical failure point if present in an area of the part that receives substantial stress, such as the head of a screw. In this case, designing a strengthening boss feature around the hole is a good precaution, or just skip the hole entirely and drill it afterwards.



Surface Imperfections and Finishing Options

If you select a PM-F0 non-cosmetic finish on a tool, the finished part will likely show small, circular, end-mill marks and tool transition lines. If you need a surface finish that's more cosmetically appealing, it's generally a simple, if more expensive, matter to manually polish the tool. A PM-F1 finish removes most tool marks, while an SPI-A2 will be smoother than a fresh jar of peanut butter. Texturing via bead blasting is another option, which generally leaves a uniform matte finish, except in thicker areas, around knit lines, and in darker materials. Bear in mind that deep slots and cavities are difficult to reach for polishing and texturing, and that fine finishes may impact quick turnaround time because of the additional effort needed for polishing. We offer eight surface finish options to choose from for injection-molded parts. In addition, we offer a number of injection molding finishing options such as mold texturing and part marking (pad printing, laser engraving).



These examples of vestiges are usually caused by the removal of the gate after molding and are mostly unavoidable, but can be hidden by orienting the part in the mold so that the cosmetic surfaces are unaffected.

Vestiges in Molding

Gate vestige is that small ugly spot at one end of the part left by removal of the gate after molding, usually with a side cutter or razor knife. It's an unavoidable fact of injection molding. The only thing, for the most part, that can be done to avoid it is orienting the part in the mold such that cosmetic surfaces are unaffected—when molding a Statue of Liberty replica, for example, the gate should be placed on the soles of Lady Liberty's feet. When submitting a design to Protolabs, always be sure to speak to an applications engineer to be sure surfaces that require a vestige-free appearance can be accommodated. We may have options to change a gate style depending on the material and part geometry. It is much easier to do this during the review stages rather than after the mold design stages have begun.

Jets, Orange Peels, and Other Molding Issues

There are several miscellaneous problems that can crop up with injection molding, several of which can be tied back to wall thicknesses that exceed general recommendations:

- Jetting, a wormlike swirl that appears near especially thick gate areas, is caused by temperature variations within the material flow.
- Similarly, a surface that looks like an orange peel can be caused by flow variations in the mold cavity, usually in thicker sections of the part.
- Silvery streaks and material flaking is known as splay, and can occur as a result of moist or degraded resin, but can also be caused by material shear due to higher than normal injector-screw speeds.
- Blush, a cloudy discoloration normally found near gate areas, can be caused by improper fill speeds, but proper part geometry and gate placement also play a factor.

Thankfully, most of these issues can be resolved through slight modifications to part design and/or selecting a different material.

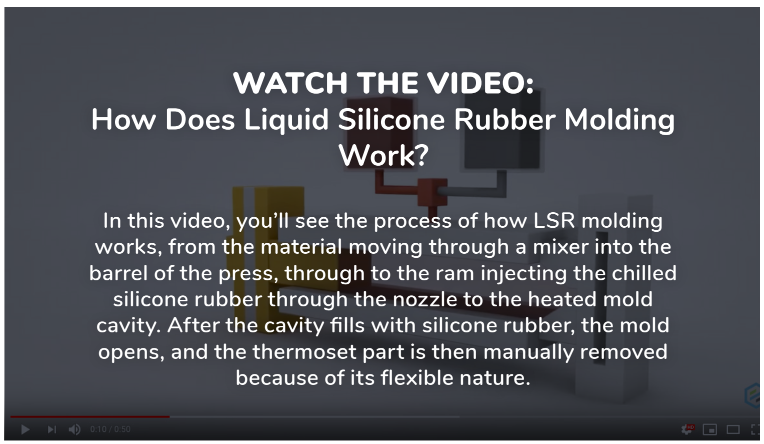
WANT MORE?
CLICK HERE FOR THE FULL ARTICLE

Designing Liquid Silicone Rubber Prototypes and Components

Learn how to optimize LSR parts with 6 simple design considerations

WE OFFER liquid silicone rubber (LSR) among our list of injection molding capabilities. LSR molding shares many similarities with conventional injection molding, but there are a few notable differences. Unlike thermoplastic resin, which is melted before injection, LSR is a two-part thermoset compound that is chilled, before being injected into a heated mold and ultimately cured into a final part. Since LSR is a thermosetting polymer, its molded state is permanent—once it is set, it can't be melted again like a thermoplastic.

LSR has certain inherent characteristics. It is a strong, elastic material with excellent thermal, chemical, and electrical resistance. LSR parts also maintain their physical properties at extreme temperatures and can withstand sterilization. LSR is biocompatible so it works very well for products that have skin contact. Those benefits lend themselves well to automotive, medical and food appliance industries, typically in the form of seals, gaskets, valves, and cables.



Designing parts for LSR and thermoplastics are similar, but there are some LSR-specific guidelines to consider:

COMMON APPLICATIONS FOR SILICONE RUBBER

Material properties for LSR offer strength, superior flexibility, and heat and chemical resistance. Common applications include:

- Soft-touch surfaces
- Gaskets
- Valves
- Heat insulation

LSR can also withstand sterilization and is biocompatible so it works well for products that have skin contact, lending LSR parts well to industry segments such as:

- Medical
- Automotive
- Food appliances

Dimensions

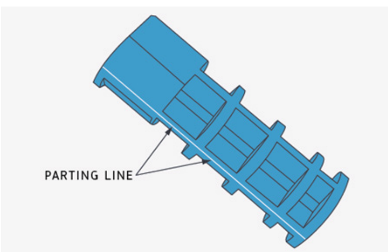
We allow for a maximum LSR part size of 12 in. (304mm) by 8 in. (203mm) by 4 in. (100mm) with depths no greater than 2 in. (50mm) from any parting line. Note that deeper parts are limited to a smaller outline. Maximum surface area is 48.4 sq. in. (312 sq. cm.) and maximum volume of part is 12.5 cu. in. (204 cc).

Wall and Rib Thickness

LSR typically fills thin wall sections with minimal challenges, and walls as thin as 0.012 in. are possible over smaller areas but down to .015 in. thick over larger areas, depending on the size of the wall and the location of adjacent thicker sections. Rib thickness should be 0.5 to 1.0 times the adjoining wall thickness. LSR is accommodating to variations in wall thickness and sink is almost nonexistent.

Shrink and Flash

The shrink rate on LSR is fairly high but is very consistent throughout the entire part with an expected tolerance of 0.025 in./in. LSR also tends to flash very easily during molding (in gaps as small as 0.0002 in.), which Protolabs helps reduce by incorporating additional features into the mold design.



Parting lines—an example is shown here in a molded part—may be encountered on plastic and liquid silicone rubber parts. Simplifying and minimizing parting lines will help you get cleaner parts faster.

Parting Lines

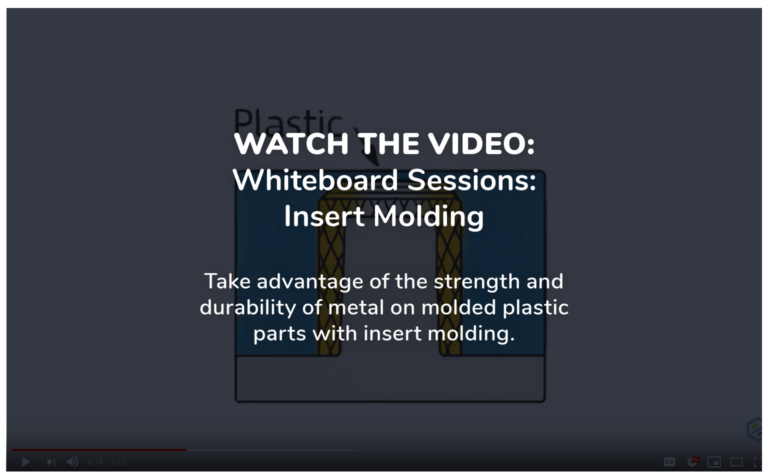
Simplifying and minimizing parting lines in your design will help you get cleaner LSR parts as quickly as possible.

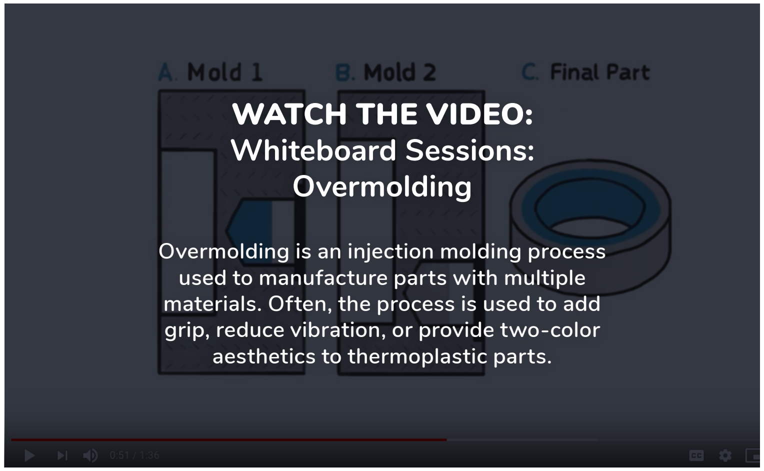
Undercuts

LSR can be molded to accommodate parts with undercuts, which are manually removed by a press operator. Mechanical tooling actions to release undercuts are selectively offered at Protolabs. LSR tends to tear fairly easily at sharp edges. In the making of interior features, it is very important to try to add fillets or radii to prevent tearing at what might be generally sharp edges, when using a manually removed undercut.

Overmolding and Insert Molding

LSR materials can also be used for overmolding and insert molding purposes. In fact, LSR materials are often used for overmolding, which allows an additional layer of resin to be added to an existing molded part to provide a combination of characteristics no single material can provide. One of the most common applications is to add a soft, functional, hand-friendly layer of rubber-like material, typically a thermoplastic elastomer, over a hard substrate. Another is to change or enhance the appearance or cosmetics of a part by overmolding material of a different color or finish to it. As an example, overmolding shows up on anything from medical devices and hand tools to toothbrushes and oven knobs. Most overmolding done on the above mentioned items are a TPE or TPU material and not silicone.





Insert molding is used to add strength and durability to parts, usually by bolstering certain pivotal points on an LSR part with metal—a brass bearing journal, for example, or a stainless steel threaded insert—or a high-temperature plastic material. Because of the high-cure temperature of LSR molding, suggested plastic materials would be Valox, PEI (Ultem) or PEEK.

There usually needs to be mechanical bonding features when insert molding LSR onto other parts, because LSR generally does not chemically bond. Additionally, Protolabs does not allow the use of a primer to make a chemical bond.

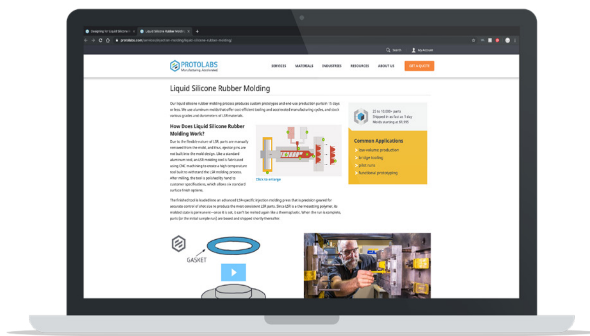
For more detailed information on mechanical and chemical bonding, see the table at the bottom of this tech tip.

Part Ejection

Ejector pins are normally not used during LSR molding due to the flashy nature of the material. Thus, parts should be designed so they can be retained on one half of the mold when it is opened at the end of the molding cycle. The part is then manually de-molded, often with air assistance.

LSR has been in the industry for a long time, but we offer LSR parts in volumes of 25 to 5,000+ in three weeks or less. To learn more about our LSR capabilities—including additional design guidelines on draft, finishes, and more—check out our LSR page. If you have a 3D CAD model ready, upload it now to get an interactive quote with design analysis and pricing information within hours.

check out the
LSR PAGE



Digital Manufacturing Helps Medtech Firm's Goal to Improve Patient Outcomes

HemoSonics used CNC machining, injection molding, and other services to quickly develop blood analysis machine

AT A GLANCE

Challenge
Slash the time of the prototyping process: Engineers at HemoSonics were iterating through designs rapidly under tight deadlines during the company's early years of R&D, and needed a contract manufacturer that could machine parts quickly. On a more recent project, HemoSonics engineers needed prototype parts about the size of a computer monitor for the casing around the Quantra System blood-clot analysis machine. The prototypes were to demonstrate form, fit, and function of the Quantra System to physicians at various hospitals.

Solution
Produce custom prototypes for several design iterations within days using quick-turn digital manufacturing. Machining was implemented for some products, and more recently, 3D printing, injection molding, and sheet metal fabrication for the Quantra System products. Engineers also used some finishing options for injection molding, including heat staking and pad printing.

Outcome
Following years of research and development that included securing key patents, conducting numerous hospital studies, and consulting with physicians and other clinicians, Quantra System products are now on the market in Europe and will launch soon in the United States.

AS OFTEN HAPPENS in the medical industry, innovative ideas, hatched in university research settings, spawn innovative companies, which create innovative products. A case in point: HemoSonics.

The Charlottesville, Virginia-based medical device company was started in 2005 by two professors and a post-doctoral research student at the University of Virginia School of Medicine's Bio-Medical Engineering program—Bill Walker, Mike Lawrence, and Francesco Viola respectively. The trio identified a method for measuring the stiffness of blood clots by using ultra-sound imaging technology, and created a system built around that technology aimed to improve patient outcomes and reduce costs.

A number of years followed of extensive research and development, which included securing key patents, conducting numerous hospital studies, and consulting with physicians and other clinicians.

More recently, HemoSonics has been prepping to bring its Quantra System diagnostic products to market, with prototyping and end-use manufacturing help from Protolabs. In fact, Protolabs has worked with HemoSonics since 2011, from

"Speed and flexibility—being able to deploy different manufacturing options—and a commitment to customer service, are the main reasons we use Protolabs."

those early R&D days to more recent, end-use production work on the Quantra System.

HemoSonics successfully launched its products commercially last year in Europe, and hopes to enter the U.S. market soon. HemoSonics has expanded its offices into Durham, North Carolina, and has grown to 50-plus employees.

A Need for Speed and Flexibility, Solved by an Agile Supplier

In HemoSonics' early research and development days, engineers were "iterating through multiple designs under tight deadlines," said Andy Homyk, senior engineer, who joined the company more than six years ago when it had just five employees. "We were on a tight timeline so we needed a supplier who could machine parts for us quickly, within a couple of days."

A number of suppliers contacted could not meet those challenging deadlines. Protolabs could, Homyk remembered. "The difference in lead times was dramatic."

That was in 2012. Since that time, Protolabs has produced hundreds of prototypes and thousands of components for HemoSonics, using 3D printing, CNC machining, injection molding, and sheet metal fabrication for a variety of projects and parts: robotic fixturing, thermal control units, pneumatic manifolds, and more.

"Speed and flexibility—being able to deploy different manufacturing options—and a commitment to customer service, are the main reasons we use Protolabs," Homyk said.

More recently, HemoSonics looked to Protolabs for help with the "skins" or casings that fit around the Quantra System, Homyk explained. HemoSonics engineers needed to design prototypes about the size of a computer monitor—first using 3D printing and then injection molding—to demonstrate form, fit, and function of the Quantra System to physicians at various hospitals.

Protolabs produced the casing or shell for HemoSonics' Quantra System blood-clot analysis machine.

The Quantra System is designed as a rapid, easy-to-use diagnostic platform that uses disposable cartridges to conduct a panel of tests. The Quantra Hemostasis Analyzer is designed for use in critical care settings that require results to be generated quickly from an instrument that is easy to operate at the point of care.

A challenge emerged when the project switched from additive manufacturing to injection molding. "These are pretty big parts, so one of the molding challenges, in prototyping, was color matching," Homyk said.

Digital Manufacturing Helps Medtech Firm's Goal to Improve Patient Outcomes | continued

The Quantra System blood-clot analysis machine measures the stiffness of blood clots by using ultra-sound imaging technology, and is intended to improve patient outcomes in critical care settings such as operating rooms.

Molding Materials and Finishing Touches

HemoSonics wanted these casings Pantone color-matched to its marketing department's specifications. One of the ways Protolabs normally does that, in the injection molding process, is to take the plastic resin in the natural color of the specific material chosen, apply around a 3 percent salt-and-pepper mix of colored resins, and final parts are typically very close to the preferred color. But, because of the nature of HemoSonics parts, some swirling and flow marks were showing up on them. "The first batch of parts did not look good cosmetically," Homyk remembered.

Undaunted, Protolabs went to one of its plastic resin suppliers and the supplier collaborated with Protolabs and HemoSonics to pre-compound the colors. "They mixed the plastic with the dye before molding to get pellets with a nice uniform color," Homyk said. This custom, pre-colored resin produced flawless parts. "This again speaks to Protolabs' customer service going the extra mile, and to how agile the company can be."

Material selection was also carefully considered, Homyk said, given that a requirement of almost any kind of medical device is that it needs to meet certain flammability standards. For the casings, HemoSonics opted for an ABS plastic that met those standards and also offered durability.

Beyond machining, 3D printing, and injection molding, HemoSonics engineers also used some additional finishing options on the injection-molded parts, such as heat staking and pad printing. Heat staking is a process that uses a heated stake to melt metal threaded inserts into plastic parts. This makes it so that screws can be used to attach the Quantra casing parts to a frame, for example.

Pad printing is a process that uses a stamp called a cliche to apply colored logos or decals to parts. HemoSonics used pad printing to put company logos on the Quantra System case parts. Protolabs plans to make these and other finishing options such as mold texturing and part assembly more widely available in the future.

The Outcome? Quantra System Launched in Europe, and is Progressing toward the U.S.

Those long years of research and development, multiple design iterations and prototypes, numerous hospital studies, scores of visits to physicians and other clinicians, the securing of key patents, and the landing of important certifications in Europe—including the CE Mark, is finally paying off, Homyk said. Last year, the Quantra System launched in Europe, and company leaders hope to launch the Quantra System in the U.S.

Going forward, Homyk expects Protolabs to continue to play a key supplier role to support the company's work. "We pick you guys (Protolabs) because of familiarity, speed of production, flexibility, and exceptional customer service."

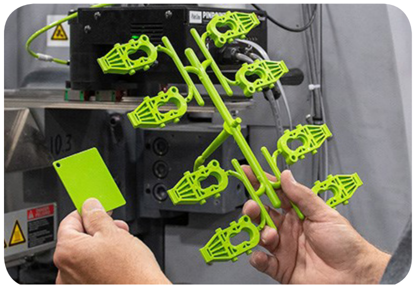
***NOTE: In March, HemoSonics announced that its Quantra Hemostasis Analyzer platform, and its initial QPlus cartridge, have been granted marketing authorization by the FDA and are now available and on the market in the United States.*

Finishing Options for Injection Molding

Choose from a wide selection of finishing options for molding that strengthen parts, improve cosmetic appearance, provide customization, and other benefits.

Mold texturing involves applying industry standard textures to a mold. With mold texturing you can expect the equivalent of a Mold-Tech finish.

Custom colorant matching is offered on customer-supplied Pantone numbers and customer-supplied part samples. The color matching process is simple and fast. Note that metallics, neons, and brighteners are not accepted at this time.



Molded sample plaques are matched with actual parts to confirm custom color match is accurate.

Threaded inserts are possible through secondary heat staking and ultrasonic welding processes. Heat staking is the process of heating threaded inserts and press fitting those inserts into molded part geometries. Keep in mind that we are unable to accept custom inserts at this time. Ultrasonic welding uses high frequency acoustic vibrations to press fit inserts into molded part geometries. Like heat staking, ultrasonic welding is currently open to threaded inserts, but not custom insert.

Pad printing transfers a two-dimensional image onto a three-dimensional object. We will print designed images like company logos on molded parts at Protolabs. All images are reviewed for size, color, and complexity restrictions, and images for printing should be provided with locations clearly marked in reference to part geometry. PM-TI and MT11010 are the most aggressive textures we can pad print.

Part assembly helps streamline supply chains by keeping manufacturing and product assembly with one supplier.

Laser engraving is applied to the mold or directly to final parts for information such as logos or part numbers. Laser engraving ensures crisp, consistent information on each part or serialization. Acetal materials are not suitable for laser engraving.

Basic assembly at Protolabs includes fastening molded parts together that we've manufactured and/or applying of labels to individually bagged parts. We do not accept customer-supplied parts for assembly at this time.

JOIN THE DISCUSSION

Email article ideas, cool projects or great design stories to the editor at marketing@protolabs.com.

protolabs.com
customerservice@protolabs.com
877.479.3680

USA
Global Headquarters
Proto Labs, I nc.
5540 Pioneer Creek Dr.
Maple Plain, MN 55359
United States

877.479.3680
www.protolabs.com

Japan
Headquarters
2-10-8 Hironodai
Zama-shi, Kanagawa
252-0012 Japan

+81.46.203.9100
www.protolabs.co.jp

Europe
Headquarters
Proto Labs, Ltd.
Halesfield 8, Telford
Shropshire
TF7 4QN
United Kingdom

P: +44 (0) 1952 683047
www.protolabs.co.uk