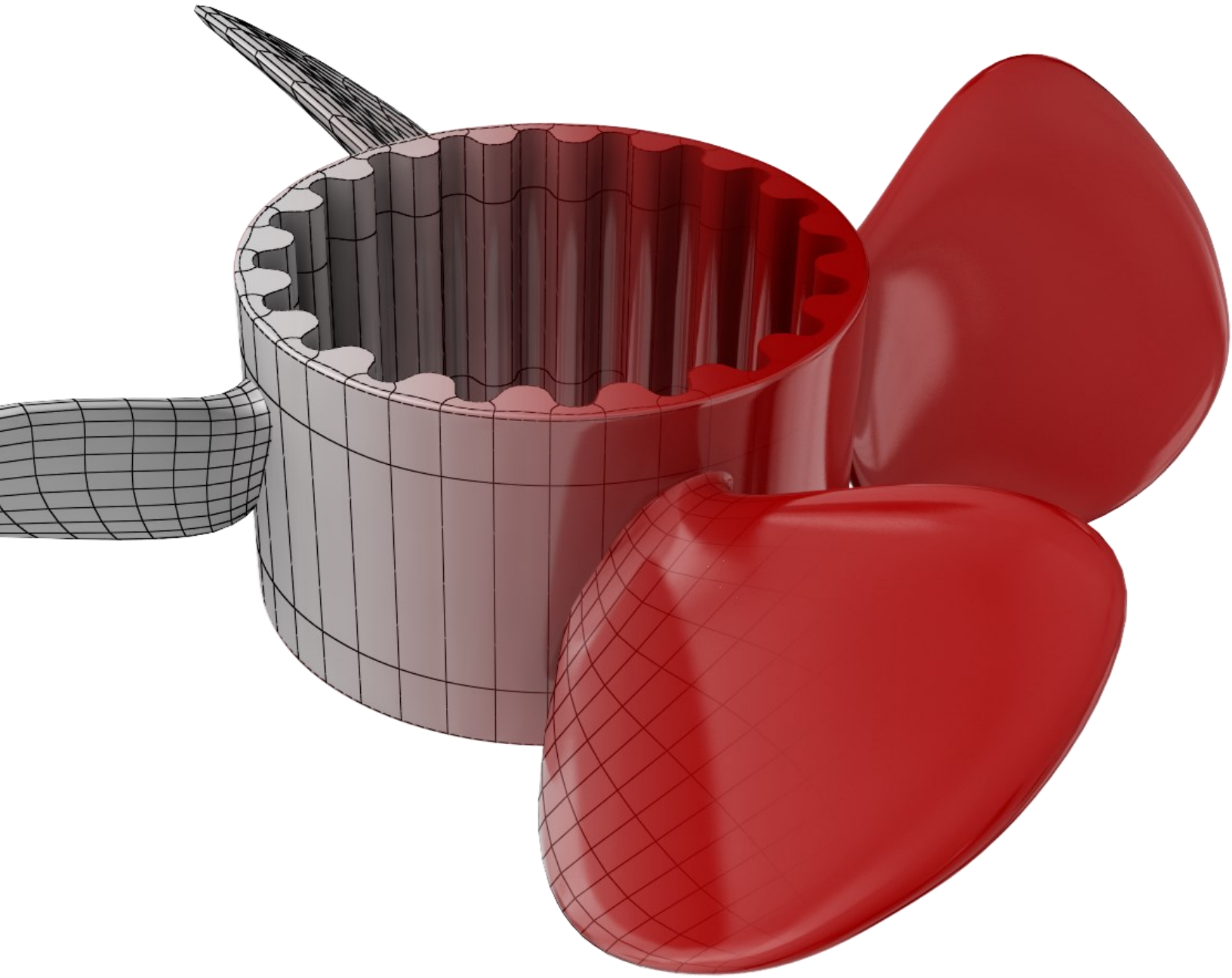
Design Guidelines for Injection Molding



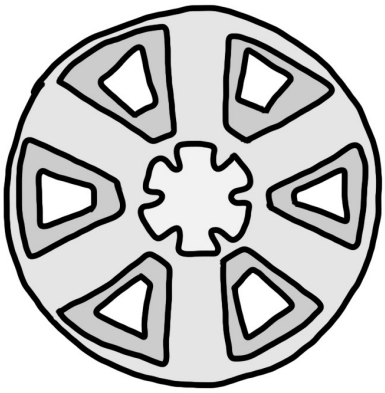
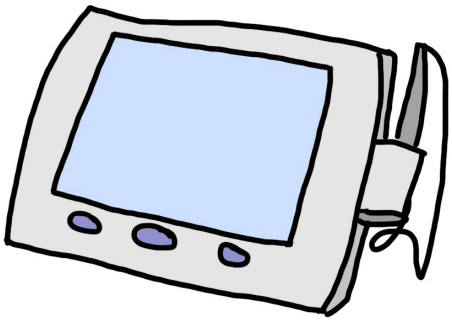
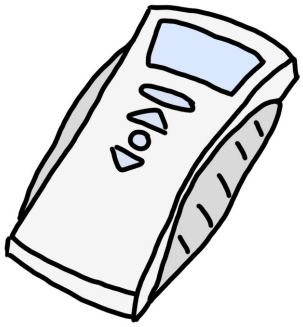
REVPART

TABLE OF CONTENTS

INTRODUCTION TO INJECTION MOLDING

POWER TIP:

Selecting a plastic resin is an important first step. When molds are engineered, they are made for a specific type of plastic. Each type of plastic shrinks as it cools, but they all shrink differently. Once a mold is complete, it's only good for parts made from plastics with similar shrink rates, so a finished mold limits the kinds of plastic you can use.

THE APPLICATIONS OF PLASTIC INJECTION MOLDING

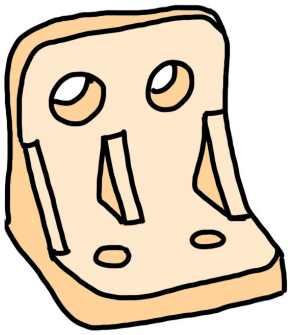
Plastic injection molding is the most common way to mass produce parts. Injection molding is great if you want to produce the same item over and over again. There are two main reasons why it is so popular: first, after an initial investment in a mold, it is the lowest cost par part; second, since the part comes out of a fixed mold, the parts are very repeatable with very accurate results. There are other reasons for its popularity as well: minimal scrap, low energy usage, and limited need for post processing.

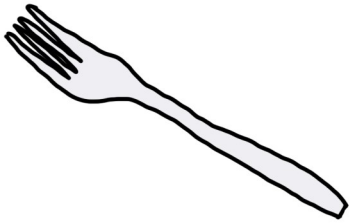
WHERE DO YOU SEE EXAMPLES?

Just about everywhere you look, across every industry, you can find injection molded parts. Some examples of injection molded parts are bottle caps, airplane windows, car dash knobs, cell phone cases, and coffee makers.

THE MAIN DISADVANTAGE

The main disadvantage with plastic injection molding is the initial cost and time to build a mold. Molds are typically built from tool grade steel and the time and effort to produce a mold are great.

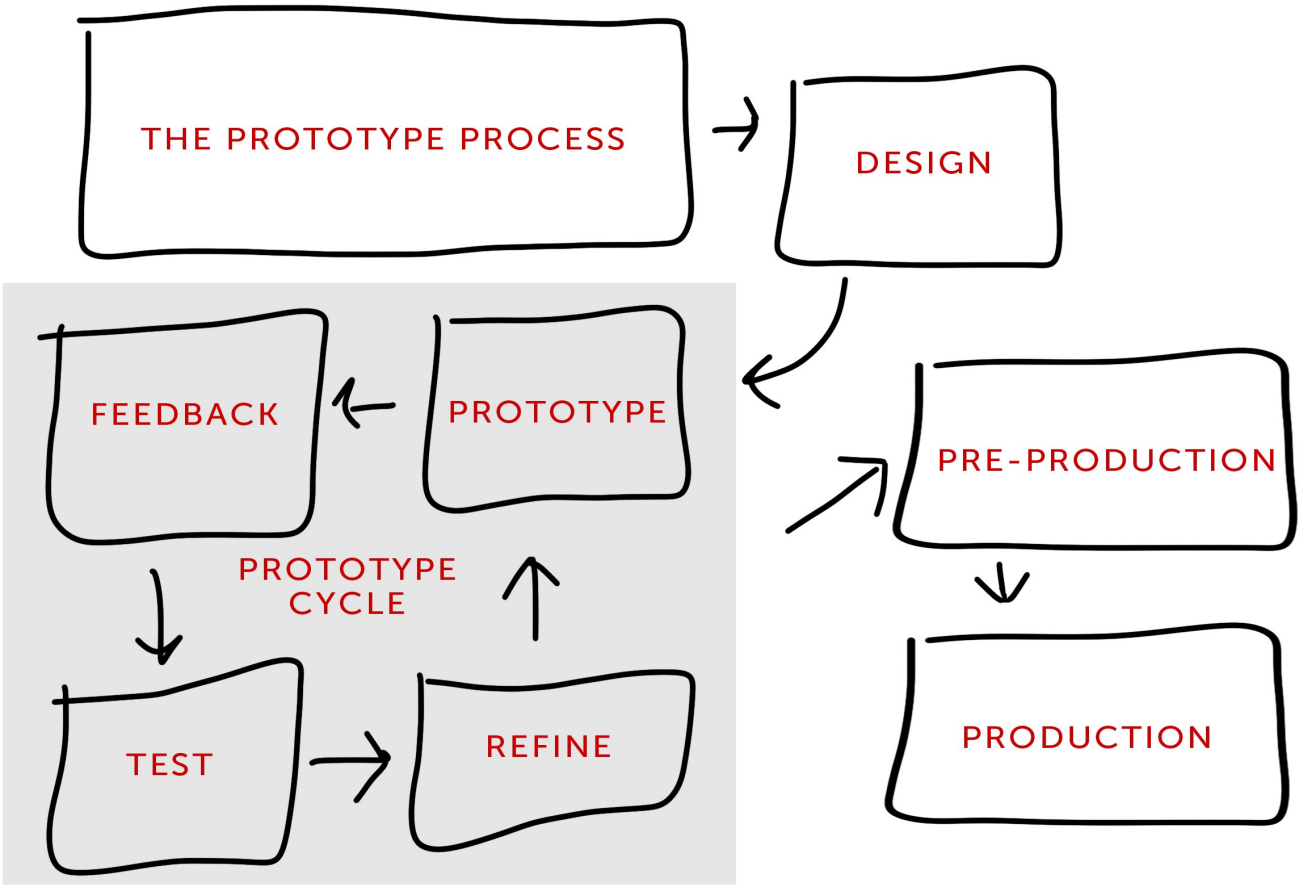




THE IMPORTANCE OF PROTOTYPING PLASTIC INJECTION MOLDED PARTS

Since the main disadvantage to plastic injection molding is the upfront mold cost, it makes sense that when it's time to purchase a mold you want to make sure to get it right the first time. Making a mold and having to scrap it can be extremely costly. That almost never happens, though, because we encourage customers to fully prototype the parts to verify the design before moving ahead with production.

It's a lot less expensive to verify your design with a prototype or two than to throw away a mold.



TYPES OF PROTOTYPES FOR INJECTION MOLDING

There are three main ways to prototype a plastic part and RevPart offers all three. These include additive prototyping (3D printing), subtractive prototyping (CNC Machining), and prototype molding.

3D PRINTING

ADVANTAGES

- Low cost
- Quick turnaround
- Make complex geometry

DISADVANTAGES

- Limited functionality
- Typically not used for production
- May not be suitable for complex geometry

CNC MACHINING

ADVANTAGES

- Use production grade plastic
- Quick turnaround
- Functional parts

DISADVANTAGES

- Some geometry constraints
- Medium cost

PROTOTYPE MOLDING

ADVANTAGES

- Least expensive per-part cost
- Final production grade part
- More finishing and decoration options

DISADVANTAGES

- Mold build time
- Mold cost

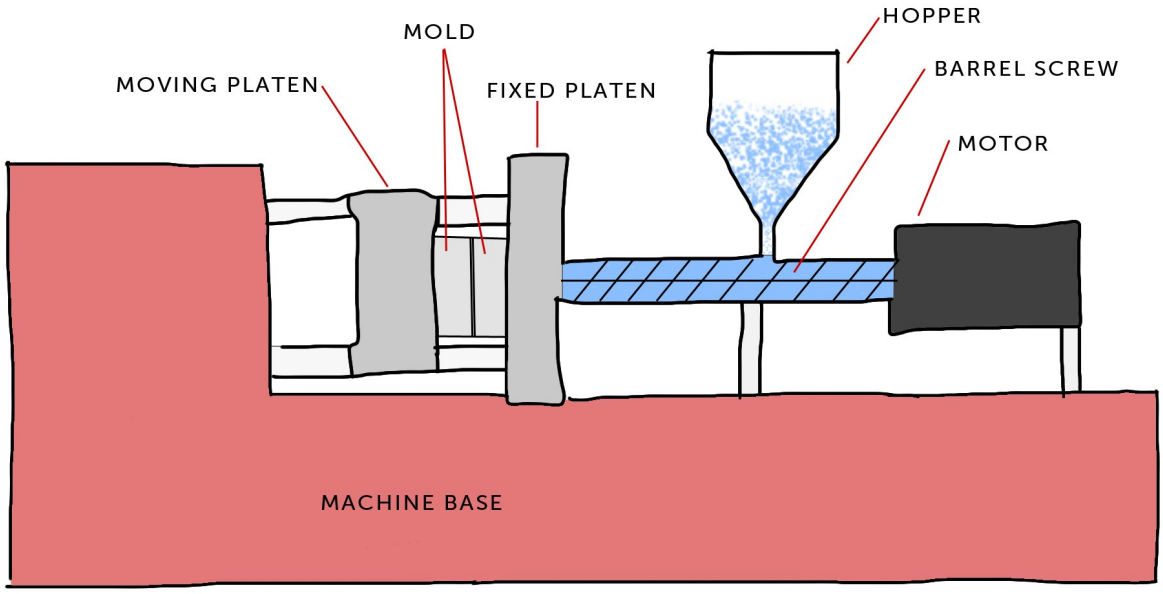
INJECTION MOLDING BASICS

POWER TIP:

Design your part just as you would for injection molding—even if you're starting out with 3D printing. This will help ensure a good part when it's time to move to production.

PURPOSE OF INJECTION MOLDING MACHINES

The simplified purpose of an injection molding machine is to hold two halves of a metal mold together, deliver a measured amount of pressurized molten plastic to the cavity chamber, and to do this as quickly and precisely as possible.



KEY INJECTION MOLDING MACHINE FEATURES

- Hopper – This is the funnel where the plastic pellets are poured.
- Barrel screw – This is where the plastic is melted and pressurized.
- Motor – This is what turns the screw to pressurize and deliver the plastic.
- Injection nozzle – This interfaces with the mold and controls the amount of plastic delivered.
- Moving platen – This is where the mold is mounted. These move back and forth and can clamp with tons of force.

TYPES OF MACHINES

Injection molding machines are sized by how many tons of force they can apply to the mold halves. Typical sizes range from 100 tons — 500 ton machines. The size of the machine needed is determined by the size of the plastic part. The bigger the part, the more force is required to help the two mold halves pushed together.

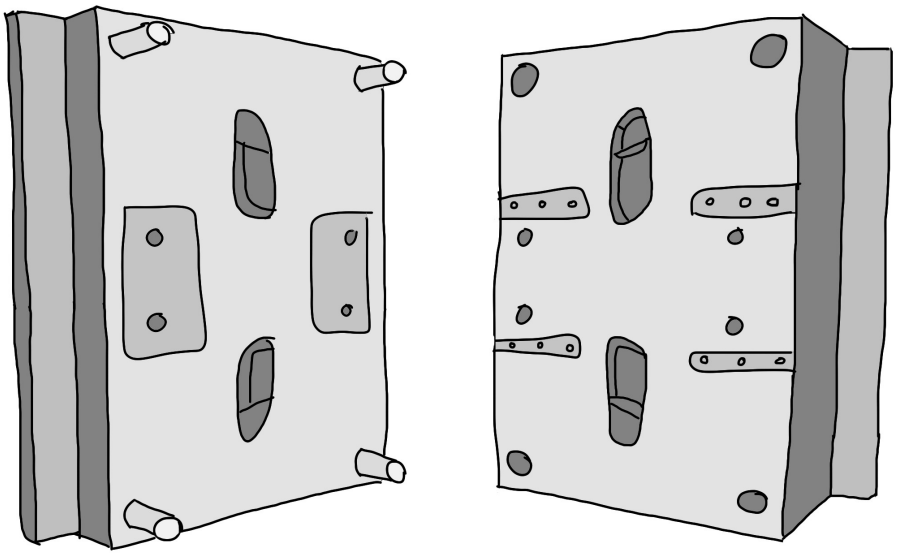
ANATOMY

Ejector pins – These are sliding pins that push the part out of the mold after it opens.

Gate – This is the areas where the plastic enters the mold and flows to the cavity.

Vent – This is the area in which the gas is released from the mold as the plastic flows in.

Slide – Slides are moving parts that allow complex part geometry. The part is molded then they slide out of the way so the part can be ejected.



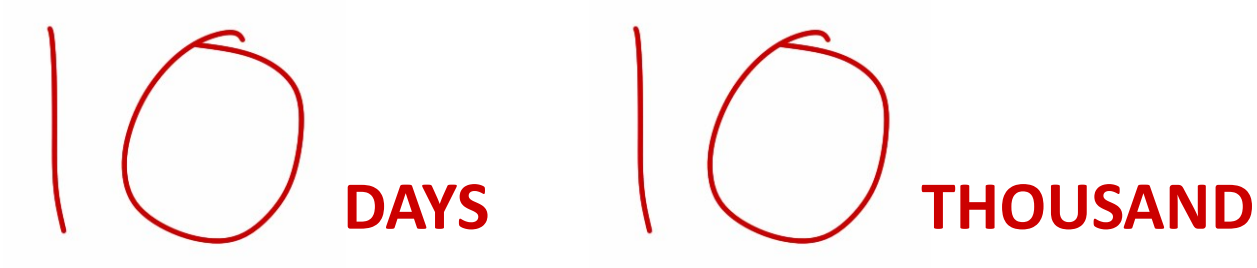
MOLDS

Molds are big pieces of metal split in half with a hollow cavity in the shape of the part you're trying to create. Molds are very beefy and contain a lot of metal because they have to withstand the high injection pressure, they have to withstand the machine clamping pressure, and they are expected to make a large number of parts.

HOW MOLDS ARE MADE

For very simple cavities, the mold can be CNC machined. Molds that require greater complexity are made by using an EDM (Electrical Discharge Machining) process. EDM uses a machined electrode to eat away the metal layer by layer. This process is slow but results in a very accurate and high quality cavity.

RevPart has developed a proprietary process which allows us to produce a prototype mold in ten days that is good for ten thousand shots.



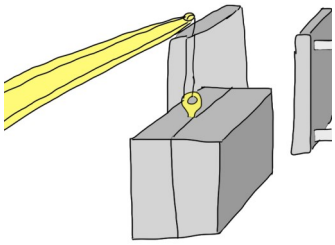
SELECTING A PLASTIC RESIN

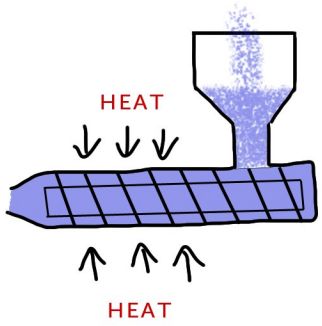
Selecting a plastic resin is always a difficult task. With over 10,000 different types available, the options can be a little overwhelming. Luckily, plastic resin falls into categories that have similar properties. You can compare the pros and cons of each plastic category, and once you find a suitable category, narrow it down to which specific resin will work for your application.

						
	Polyethylene terephthalate (PET)	High density polyethylene	Polyvinyl Chloride (PVC)	Low Density Polyethylene	Polypropylene	Polystyrene (PS)
Clarity	Clear	Hazy translu-cent	Clear	Translucent	Translucent	Clear
Moisture Barrier (MTVR)	Good	Excellent	Good	Very Good	Excellent	Poor
Oxygen Barrier	Good	Poor	Good	Poor	Poor	Poor
Distortion Temperature	155°F	160°F	150°F	110°F	200°F	170°F
Rigidity	High	Moderate	High	Low	Moderate	High
Stress Crack Resistance	Excellent	Fair	Excellent	Good	Excellent	Fair
Cold Resistance	Good	Excellent	Fair	Excellent	Poor	Poor
Impact Resistance	Good	Excellent	Good	Excellent	Fair	Poor
Alcohol Resistance	Good	Good	Excellent	Good	Good	Fair
Alkalis Resistance	Poor	Good	Excellent	Good	Good	Fair
Solvent Resistance	Good	Poor	Good	Poor	Poor	Poor
Oil Resistance	Fair	Good	Good	Good	Good	Poor
Acid Resistance	Fair	Good	Good	Good	Good	Fair

INITIAL SETUP

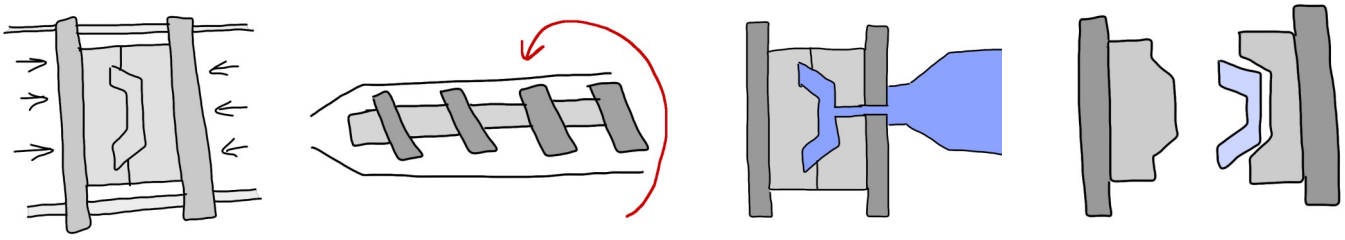
1. The mold is hoisted and loaded into the molding machine.





2. Plastic pellets are poured into the hopper on the machine. The pellets are then gravity-fed into a heated barrel where they are melted.

MOLDING CYCLE



1. The mold closes and is clamped tight with several tons of force.
2. The plastic is pressurized in the injection barrel with a turning screw.
3. The molten plastic flows at high speed and pressure though the injection nozzle into the cavity.
4. The part is allowed to cool slightly and then the mold opens to allow for part removal.
5. The process is repeated until we have the number of parts you want.

POST MOLDING PROCESS

1. Any excess plastic from plastic runs is cut and trimmed.
2. Any post processing, such as laser etching and painting, is done.
3. The parts are packaged for shipping.

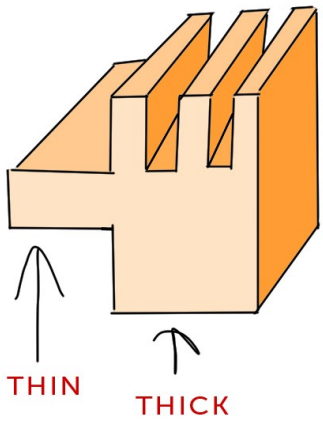
INJECTION MOLDING DETAILS

POWER TIP:

When trying to choose between 3D printing and CNC machining for prototyping, consider where you are in the design process. If at an early stage, you may want to start with a 3D print, but if you're fairly confident in your design, a CNC machined part will give you the best representation.

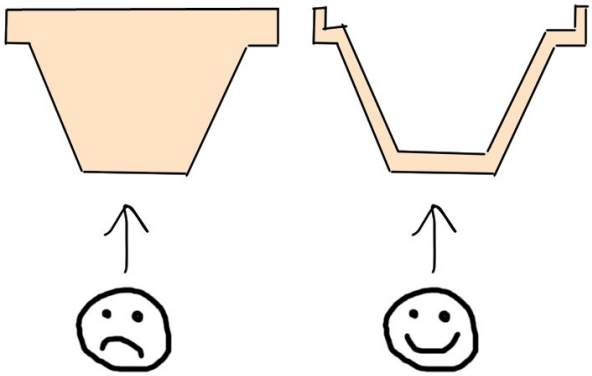
WHY IS WALL THICKNESS IMPORTANT?

The wall thickness you select determines the amount of plastic that will flow through the cavity. Wall thickness is the most important feature to consider when designing your part. Cost, quality, and production speed can all be effected by the wall thickness you select. In general, your goal should be to use the thinnest wall possible. Using thinner walls uses less material and takes less time to cool in the machine, reducing cost.



UNIFORM WALL THICKNESS

When you have a design with varying thicknesses, it can lead to quality problems such as warping. The thin walls cool before the thick wall, and the already cooled areas pull on the warm ones, causing warp. To avoid this issue, it is critical to keep the walls at a uniform thickness. If varying thicknesses are needed, make the transition as gradual as possible to avoid cooling differences.



HIGH SPEED MACHINES

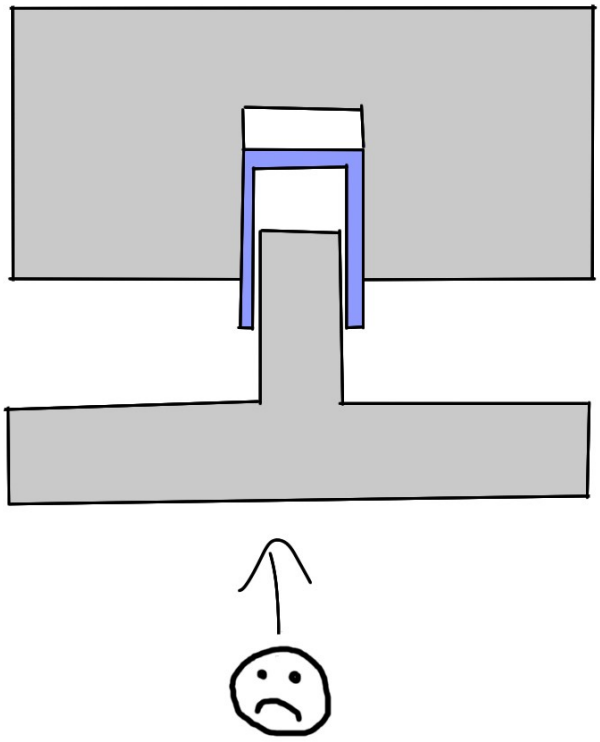
RevPart offers a variety of specialized machines to make your ideas possible. One of the specialized machines we offer is a high speed injection molding machine. This machine shoots plastic at a very high rate allowing us to mold features down to .3mm thin and 12mm thick.

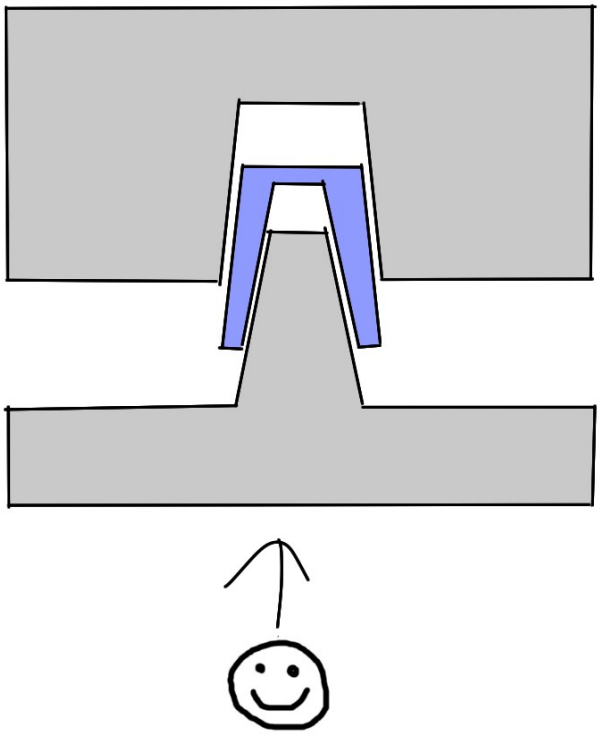
RECCOMENDED WALL THICKNESS

PLASTIC FAMILY	RECOMMENDED WALL THICKNESS (in.)
ABS	0.040 - 0.150
Acetal (Delrin)	0.025 - 0.125
Acrylic	0.025 - 0.160
Nylon	0.025 - 0.110
Polycarbonate (PC)	0.035 - 0.150
Polyester (PET)	0.025 - 0.125
Polyethylene (PE)	0.030 - 0.200
Polypropylene (PP)	0.025 - 0.150
PEEK	0.025 - 1.500

WHY IS ADDING DRAFT IMPORTANT?

Adding draft (or taper) to a part ensures that it can be removed from the mold during the ejection phase. Without draft, the part would remain in the cavity with no way of removing it safely. The amount of draft needed depends on the depth of the draw and the texture on the surface.





TEXTURE AND DRAFT

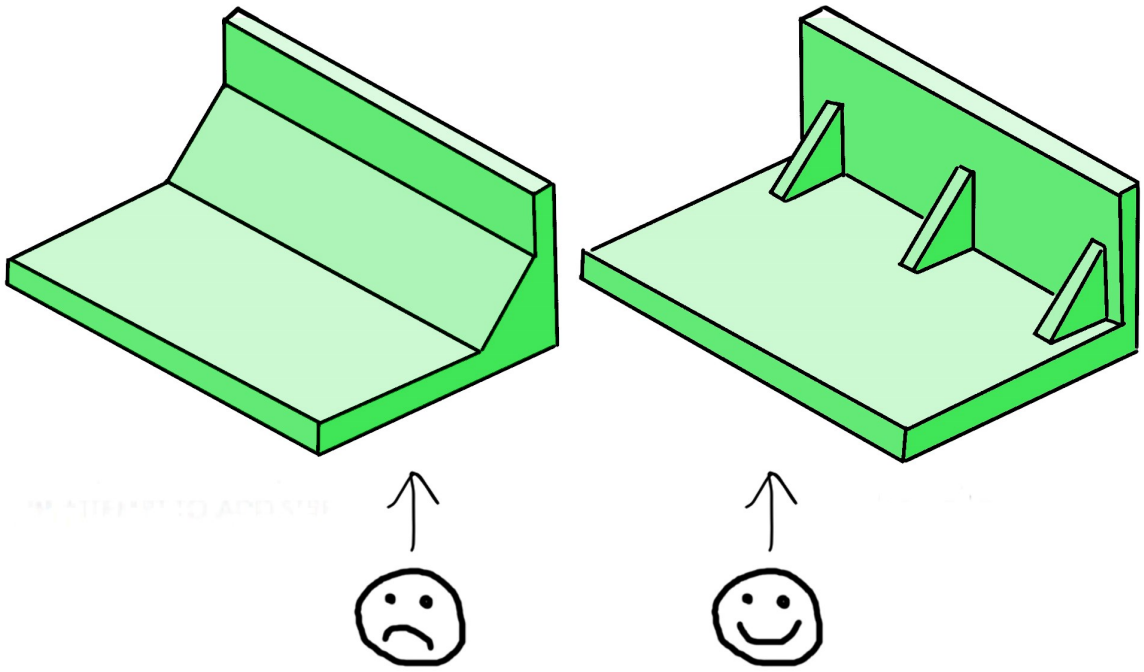
As the depth of the texture increases, more draft needs to be added so the part will eject properly and leave an aesthetically pleasing finish behind without pull or scrape marks.

RECOMMENDED DRAFT ANGLES

The typical draft angle is between 1 –3 degrees with 1.5 degrees being the most common. Draft angle can vary depending on part geometry and surface texture.
It is recommended to start with a 1.5 degree draft and let your engineer identify areas that need a larger draft angle.

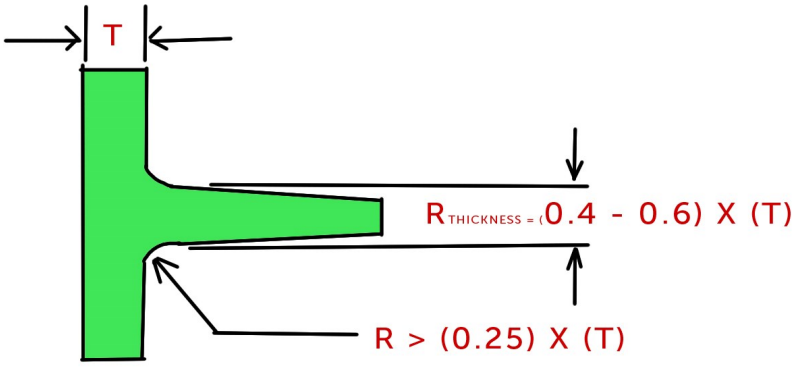
WHY ARE RIBS USED?

Ribs are commonly used in plastic injection molding to reinforce sections and add stiffness. The use of ribs is often necessary due to the uniform wall thickness requirement. You can't have thick sections to stiffen a part, but you can add several ribs in order to accomplish the same goal.



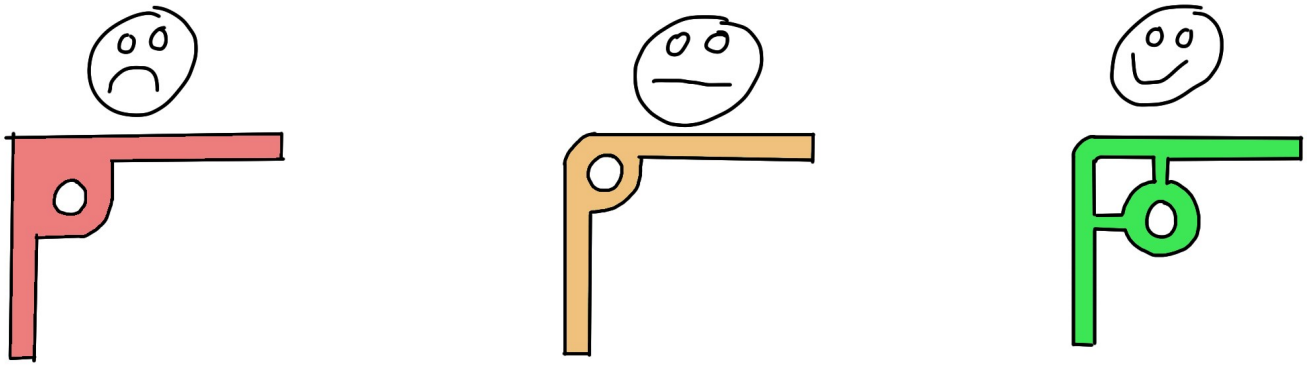
RIB THICKNESS

Just like the rest of your part, ribs should have draft in order to release properly from the mold. It is important to keep the base of the rib thickness 40%-60% of the part thickness. Thick ribs can cause sinking—a cosmetic defect. Often, it is better to use many small ribs distributed along the part than it is to use larger ribs.



WHAT ARE BOSSES USED FOR?

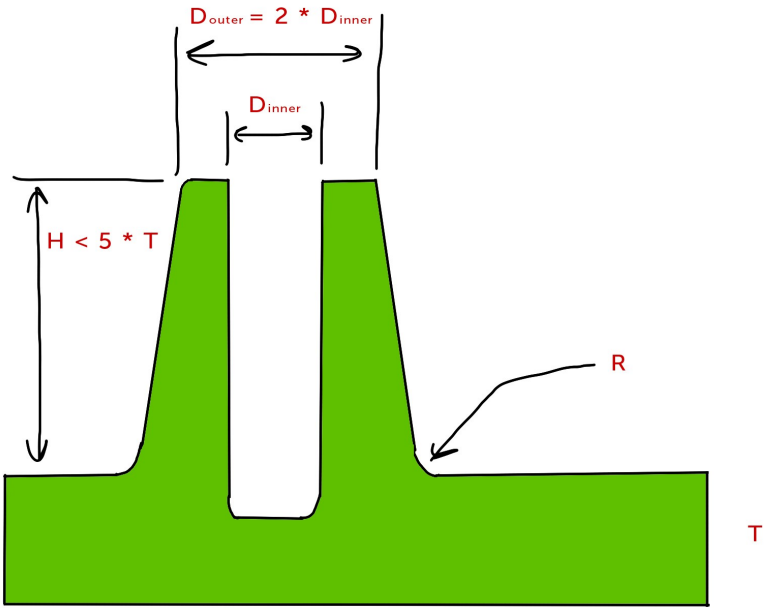
Bosses are used for attaching a fastener or registering a mating part. They are often used for mounting things like a PCB (printed circuit board) or attaching one plastic piece to another. If bosses are to take a threaded fastener like a screw, they often have a metal insert molded. Another common thing to do with bosses is to use self-tapping fasteners.



BOSS THICKNESS AND REINFORCEMENT

The following should be taken into account:

- The wall thickness of a boss should be less than 60% (at the base of the boss) of the part's wall thickness.
- The height of a boss should be less than 5 times the part wall thickness.
- Bosses should have a minimum of .5-1 degree draft.
- Bosses can be strengthened by ribs at the base.



SURFACE FINISH

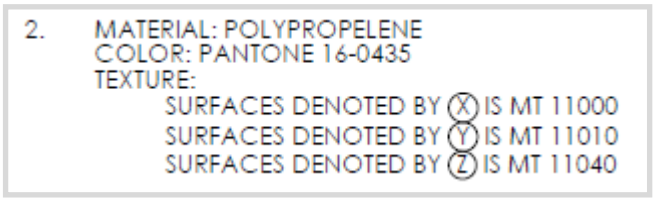
Plastic injection molded parts take on the surface of the mold. There are many options when it comes to surface finish, from a glossy polished surface to a simulated wood grain finish. The most common finishes you see are polished surfaces and textured surfaces resembling a sand blasted finish. Texture is added to the mold either by manual removal or by chemical etching.

POLISH AND TEXTURE STANDARDS

Texture is typically called out using a set of industry standards such as SPI or Mold Tech (MT) finishes. SPI finish is a requirement for mold surface finishes and is typically used for calling out the degree of polish you want on a surface. MT finishes are typically used when you want a texture on a surface.

HOW TO CALL IT OUT IN THE DRAWING

In most cases, we find the surface finish is called out in the notes of the drawing with a bubble reference pointing to the surfaces with that finish. Please see the example:

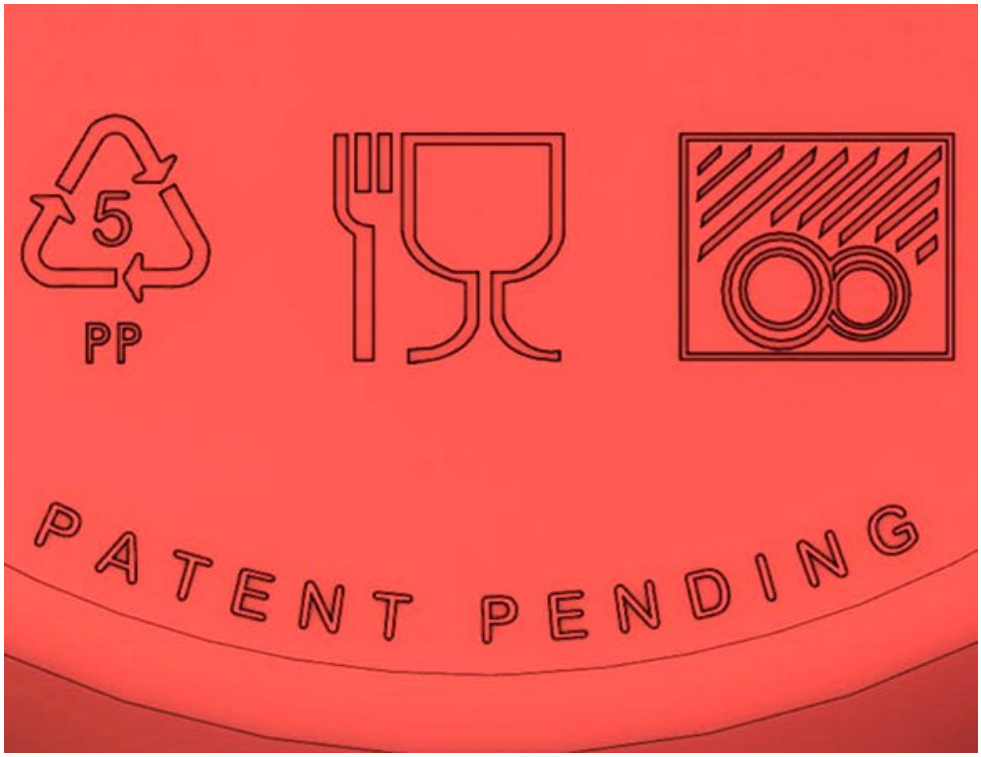


THE MOST COMMON SPI POLISH CALLOUTS

SPI A-1	GLOSSY POLISHED	#3 DIAMOND BUFF
SPI A-2	GLOSSY POLISHED	#6 DIAMOND BUFF
SPI A-3	GLOSSY, LIGHT SCRATCHES	#15 DIAMOND BUFF
SPI B-1	LOW HAZE, SCRATCHES	600 GRIT PAPER
SPI B-1	HAZY, SCRATCHES	400 GRIT PAPER

ADDING TEXT, LOGOS AND MORE TO YOUR PART

Adding information or designs to a part is very common with injection molding. Common items you see in molds are text, logos, part numbers, plastic recycle codes, cavity numbers, and date codes. Since the text is placed directly into the mold, it doesn't significantly affect the part price.



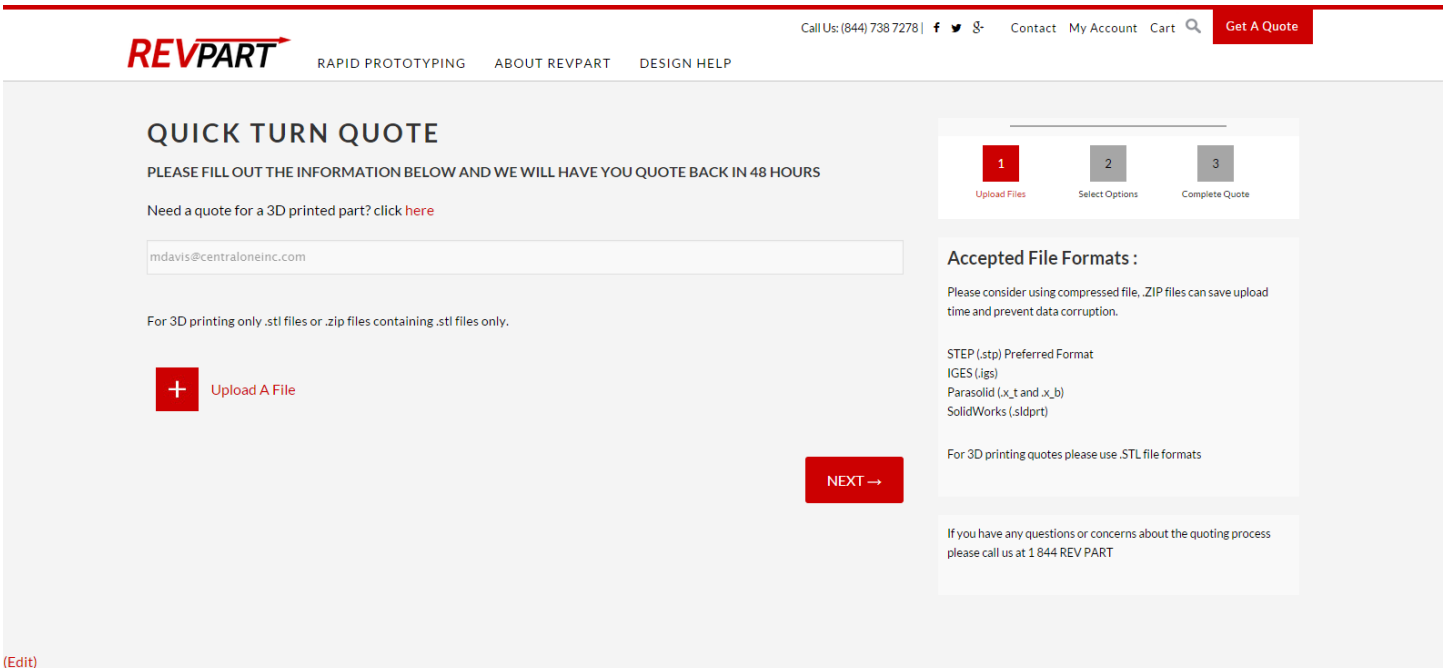
TEXT REQUIREMENTS

It is recommended that text be at least 1.5mm high and that the thickness of the text and distance between features be at least 0.5mm thick. You can choose either to emboss or deboss your text, but the text height or depth should be no more than 1 times the thickness of the letters. Please note that if you want to change the text in the future, such as patent pending, it may be a good place to ask for an insert plate. An insert plate is a removable plate in the mold that allows you to change text without the need for a tooling modification or replacement.

GETTING A QUOTE IS EASY

Thanks to our simple three step quoting process, getting a quote couldn't be easier.

GET A QUOTE TODAY



(Edit)

POWER TIP:

When adding text to your design, remember that debossed text is easier to remove because you're simply removing some metal from the mold. If you foresee needing to change your text, ask about using an insert plate for an even easier change.

DID YOU LIKE OUR BOOK?

PLEASE TAKE A MOMENT AND FOLLOW US ON SOCIAL MEDIA FOR UPDATES, COOL CONTENT, FREEBIES, EXCLUSIVE COUPONS, AND MORE....