Guttenberg Industries, Inc.

Top 10 Design Tips for Injection-Molded Parts

complete with Material Options and Tolerances

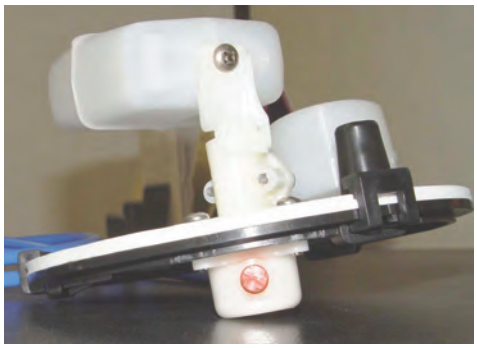


TABLE OF CONTENTS PAGE

© 2016 Guttenberg Industries, Inc.

603 S. Lincoln St., P.O. Box 70, Garnavillo, IA 52049
www.GuttenbergIndustries.com * (563) 964-1000

Introduction

Selecting a plastic-injection molding company can be a risky decision. With so many things that can go wrong, the wise manufacturer chooses a molder knowledgeable in plastic materials, technological processes, and is willing to provide engineering support.

In today's world, the injection molder must be experienced with a wide range of plastic materials, advanced mold designs and materials, scientific molding processes, and three-dimensional solid modeling with CAD/CAM capabilities. They should also maintain a state of the art quality assurance program, including high-tech inspection equipment and application of statistical process controls. Most importantly, the wise manufacturer chooses a molder who communicates actively, openly, and frequently with the customer throughout the process.

At Guttenberg Industries, we provide all of this and more. Before the tool design is even complete, our engineers work with our customers' engineers to review the part design for possible material savings, cycle time savings, and quality improvement. We advocate and apply Society for Plastics Industry (SPI) guidelines and standards for Mold Classes, Mold finish, Cosmetic Inspection, and Dimensional Tolerances.

Once the project is launched, we handle it all. We coordinate the tool build and offer weekly progress updates upon request. We work with our customers' engineers and quality professionals to ensure we understand the unique quality requirements before the mold is even complete.

When the mold is complete we sample the mold to verify the product meets the customers' requirements and tweak the mold or process until it does. We develop and document statistically proven production processes and control those processes into production. We submit samples and statistical analysis for customer approval prior to launching production. All is published in AIAG format upon request including full PPAP documentation. And if you are transferring a tool from another molder, we still do all of the same except for the tool build. Why not let us be an extension of your product development team?

Design Suggestions

Our review of part design actually begins during the quoting stage. Our experienced estimating team is skilled in detecting product features that add cost and risk downstream quality problems. If we think we can save you money, suggestions are provided during this phase.

A more detailed analysis and evaluation is also performed by our New Business Development team during the early stages of a product launch. This team actively searches for cost-saving opportunities and design features that might create a risk of problems later in the process. Simulation software is used to detect potential problems with mold flow, maximizing the potential for first-round success. Any ideas are communicated to you right away so that you may update the design quickly if you decide to do so. And, if you don't choose to act on our suggestions, that is fine, we will keep trying to help in any way we can.

Mold Design Review

Tool designs are developed by toolmakers based on the CAD models provided by Guttenberg Industries on the behalf of you, the customer. Before any actual tool work begins, the tool design must be approved by Guttenberg Industries. Improper tool design can lead to quality problems, increased production costs, and limited tool life. To minimize your risk, as well as ours, our team of experts reviews each tool design with a point checklist, ensuring the highest quality tool for your money.

Process Development and Control

Excellent product quality begins with a great tool and a robust, repeatable process. During pre-production sampling we validate the tool, but we also develop robust, repeatable processes using scientific injection molding methodologies. Scientific molding is a proven methodology for breaking the molding process into multiple distinct stages, and assembling the best combination of each stage for the best possible outcome. Application of scientific injection molding methodologies ensures a clear distinction between mold problems and processing challenges, but also serves as a basis for future statistical process control.

Capability Studies and Sample Submission

When the mold is sampled and a process is developed, representative samples are evaluated to validate both the tool and the process. A layout is performed detailing actual measurements for each dimension stated on the print you provide, while capability studies are performed on at least 30 pieces to validate statistical capability of Cpk 1.33 or greater (if specified by the customer). When all of the above criteria are met, samples and data are provided to you, the customer.

Production Approval

When your team is satisfied that all of your specifications are met and certain that the results are repeatable, we will submit sample parts and all of the measurement data to you for approval. If you are satisfied with the results, simply provide your approval and we are immediately ready for production. If not, simply discuss the issue with our team and we will resolve the problem promptly.

Product Development Lead Time

Time to market is critical today. The quicker you can get your product into the market the quicker you can begin earning returns on your investments. At Guttenberg Industries we understand this and continually strive to help you minimize this duration. Rapid prototype services are used where appropriate and quick turn-around tooling is often available upon request. Please let us know if this is something that you are interested in pursuing.

Design for Manufacturability (DFM)

Design for Manufacturability, or DFM, is a time-tested tool to help product designers minimize costs and time to market. Complex designs incur additional costs. Sometimes complexity is necessary. However, unnecessarily complex plastic injection molding designs can increase the manufacturing costs of plastic parts significantly. Our professional staff at Guttenberg Industries aims to help you use DFM in order to keep your designs as simple as possible while keeping costs to a minimum.

Guttenberg Industries is willing to share some simple DFM tips to help you maximize your profits with our Top 10 Design Tips for Injection Molding. In return, we hope that you will consider doing business with us – or at least provide us an opportunity to quote your next project.

Top 10 Design Tips for Injection Molding

#1	Understanding Draft
#2	Thicker is often Weaker
#3	Understanding Ribs and Gussets
#4	Importance of Radii
#5	Gate Location
#6	Round, Flat, Straight Part Myths
#7	Recognizing and Avoiding Undercuts
#8	Threads – Inside and Outside
#9	Consider Total Cost, Not Price
#10	Materials and Tolerances

Tip #1: Understanding Draft

Draft is an angle incorporated into the wall of a mold, thus the shape of the plastic part, so that the opening of the cavity is wider than its base. For those unfamiliar with draft, you might notice that plastic drinking cups are always wider at the mouth of the cup than at the bottom. This shape is a demonstration of draft. It is smaller at the base than at the mouth so that the cup will come out of the mold. Draft is essential for injection molding.

In very basic terms, an injection mold involves a cavity – or female half, and a core – or male half. During the injection molding process molten plastic is injected into the mold and fills in the gap between the cavity and core. As the plastic cools it solidifies and takes this shape. As the plastic cools it also shrinks, and in so doing effectively grips onto the core. Without draft, this gripping action caused by the plastic shrinking onto the core can become nearly impossible to overcome without damaging or destroying the shape of the plastic part. But with sufficient draft, the part is able to break free of the core with much less distortion and after a very short distance of travel.

From a molder's perspective, the more draft the better. Yet from a designer's perspective, draft often requires a change in shape from the original design concept. So as a designer, it is best to keep the concept of draft in mind as the design takes shape. For experienced designers of parts using plastic, it can be helpful to anticipate where the two halves of the mold will separate, known as parting line. The goal should be to shape the part to allow for draft in both directions away from parting line. If not experienced in designing for plastics, we can help anticipate draft during the early stages of design in order to avoid costly or frustrating design changes in later stages.

All three of the products to the right contain draft. The two parts on the right illustrate draft in a more obvious maner, but the part on the left also had draft designed into it so that the part exited the mold easier. As little as 1% draft can sometimes be enough.

Tip #2: Thicker is often Weaker

In materials such as wood or metal, thicker usually means stronger. But this is not necessarily the case for injection-molded plastics. When designing for injection-molded plastics, there is a maximum recommended wall thickness that tends to vary from one material to the next. A wall that is thinner than this recommended thickness tends to be weaker, but so does a wall that is thicker. Why does a thicker wall get weaker?

The answer has to do with the injection molding process and the plastic itself. As molten plastic cools, it also shrinks, layer by layer until each section is completely solid. But it cools and shrinks from the outside in, and as the outer shell shrinks inward, the center is simultaneously pulled toward the outer shell. So as the material in the center of the wall cools, it is simultaneously being pulled in two opposite directions toward the outer wall on both sides. If the wall is thick enough, the tension in the center of the wall can become so great that the material will completely pull away, leaving a vacuum void (hollow). In less severe cases, a void may not be visible, but the material in the center of the wall might still be under so much tension that the strength of the whole part is weakened.

Excessively thick walls also increase the cost of the injection-molded product in two ways. First, the thicker a product wall, the more plastic material will be required to mold it. Second, the thicker a wall, the more time it takes to cool it during the injection molding cycle. Increased cycle time means higher cost for the manufacturer and the customer. When designing injection molded products, strength is often best achieved by incorporating ribs or gussets into the product design.

Tip #3: Understanding Ribs and Gussets

A rib is a side wall or structure of side walls that are attached to the main wall, usually at right angles. Ribs add strength and rigidity to a primary wall without the dangers and high costs caused by excessive wall thickness. A gusset is a triangular-shaped rib supporting a main wall.

There are important considerations when designing ribs. Wherever the rib intersects with the main wall, that intersection will be thicker than the other sections of the main wall. If the rib is too thick, it will cause sink marks in the main wall.

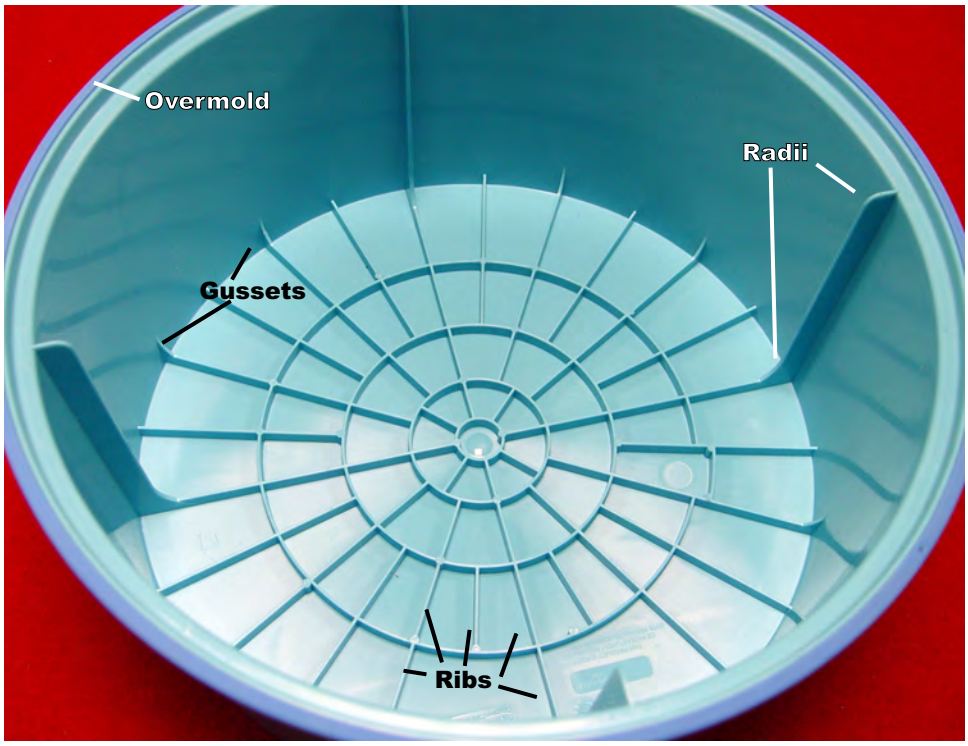
Why the sink mark? For the same reason explained in the thick wall section. As the plastic cools and shrinks, the mass of material and the intersection of the rib and the main will be so thick that it will actually collapse the main wall as it cools and shrinks. This collapsed main wall shows as a sink mark.

To avoid sink marks, a rib should never be greater than 2/3 the thickness of the main (intersecting) wall. Ribs that are taller than one inch should be even thinner, not to exceed ½ the thickness of the main (intersecting) wall.

Tip #4: Importance of Radii

During the injection molding cycle, molten plastic is injected into the space between the two halves of the mold. As the molten plastic flows into this gap, it encounters a labyrinth of metal that it must flow past. If the edges of this metal are sharp, it causes excessive sheer heat in the plastic, which can damage the plastic and weaken the molded part.

All of this is easily avoided by incorporating radii into your part design. The simple rule of thumb is "**No Sharp Edges**." If the plastic part is designed with rounded corners and radii at rib intersections, those radii will automatically transfer to the tool design, thus preventing the risks of sharp corners.



Tip #5: Gate Location(s)

The location where the molten plastic is injected into the space between the two halves of the mold is called a gate. Every injection-molded part has at least one gate; some have several gates. The type of gate and location of the gate is driven by the shape of the part and the customer's preferences for tool and piece price. Choosing the right gate for your product design can be difficult. Below are a few examples of common gate types.

Direct Sprue

As the name implies, with this style of gate the material enters the molded part directly through the sprue. A sprue is the primary channel where plastic flows from the molding machine into the mold. So in this case, the material flows directly from the molding machine, through the sprue, and into the center of the plastic part.

Pros: Compared to other gating options, this style is generally the least expensive in tooling. Also, if significant amounts of material need to flow into the tool as quickly as possible, direct sprue will likely do this best.

Cons: The location of the gate tends to be in the center of the molded part, which is not always the most desirable from a cosmetic perspective. Also, the size of a direct-sprue gate is quite large, thus unattractive when compared to other styles of gates. Sometimes the direct sprue gate even contains small vacuum voids. Finally, the leftover sprue may sometimes become a source of wasted material.

Edge Gate

With this style of gate, the molten plastic enters the molded part at the parting line of the tool. When the part ejects, a "runner" of excess plastic comes with it. The runner must then be removed from the finished part by clipping it - either by a human or through robotic automation.

Pros: The tool cost for an edge gate is lower than the other alternatives and allows for a significant volume of material to enter the mold which is important in some instances.

Cons: The cost of removing the part from the runner will affect part pricing and the runner itself may sometimes become a source of wasted material.

Three-Plate Drop Gate

Like the edge gate, a three-plate drop gate creates a runner that must be removed from the mold at the end of the molding cycle. But unlike the edge gate, the three plate drop allows material to enter the part from behind the cavity, away from parting line. In a three-plate mold, the runner also separates from the parts automatically, eliminating the cost of clipping parts from the runner.

Pros: Less labor than edge gating or direct sprue options.

Cons: Runner must still be removed from the mold either by mechanical tool action or by robotic automation. Also, the runner itself may sometimes become a source of wasted material.

Hot Manifold Gate System

With a hot manifold system, the plastic remains molten within a heated manifold inside the mold until it enters the part itself. As the molding cycle nears completion there is solid plastic on one side of the gate and molten plastic on the other side of the gate. This is also called "runnerless" molding because there is no runner to dispose of.

Pros: No labor is required to remove the parts from the runner, because there is no solid runner. Also, there is no wasted material from the runner.

Cons: This is the most expensive type of gating system to build. Changing colors during production is difficult and costly because it is difficult to "purge" the material held in the manifold in the mold. In tight tolerance molding conditions, the hot tip may not offer the most consistent, repeatable outcome possible, although the use of valve gates may help offset this problem.

Submarine and Hook Gates

Submarine (Sub) and Hook gates are very similar to the edge gate in that they create a runner that must be removed when the molding cycle ends, and this runner system tends to follow the parting line of the mold. But unlike the edge gate, submarine and hook gates tunnel through one half of the mold at the last moment and enter the part below parting line. When the parts are ejected from the mold, the gate is sheared from the runner, automatically de-gating the parts. The runner, now free of the parts, either falls into a chute attached to the molding machine, or is removed by robotic automation.

Pros: Less labor than edge gating or direct sprue options. Also, it cost less than the three-plate option.

Cons: Runner must still be removed from the mold either by mechanical tool action or by robotic automation. Also, the runner itself may sometimes become a source of wasted material.

The image at the top left illustrates the very small gate marks on the parts (top left) and the manner that the parts are alligned in the mold and the bottom image shows the runner and the small gates. The photo above illustrates that the gates are located in an area that the consumer will not see when theis part is assembled with the rest of the product.

Valve Gate

The valve gate adds a valve rod to the hot runner gate. The valve can be activated to close the gate just before the material near the gate solidifies. This allows a larger gate diameter and smoothes over the gate scar. Since the valve rod controls the packing cycle, better control of the packing cycle is maintained with more consistent quality.

Pros: These valve gates typically allow for shorter cycle times in addition to better part surface at the gate location. This is a preferred method for thin-wall parts, foam and gas assist injection

Cons: Requires equipment with technology above base levels and personnel with advanced technology and engineering education.

Tip #6: Myths about Round, Flat and Straight Parts

Injection-molded plastic parts are not, and cannot be, perfectly round, flat or straight. It may seem funny to read, but it is true. When it comes to injection-molded plastic products, roundness, flatness, and wall straightness are more an ideal than a reality. The problem with these feature types in injection-molded plastic involve characteristics such as gate location, part geometry, and molecular and fiber orientation.

Gate Location and Pressure Loss: Plastic pressure inside the mold changes in relation to the gate location. Pressure is at its greatest near the gate, and at its least at the furthest point from the gate. Plastic pressure inside the mold affects part shrinkage, so the further you get from the gate, the more shrinkage you tend to see. Thus, round features further from the gate shrink more than similar features near the gate, creating in ellipse instead of a circle. This same phenomenon can also cause flat parts and straight edges to warp.

Part Geometry: The shape of your design also affects roundness, flatness, and straightness. This is again due to shrinkage issues. In general, thick areas tend to shrink more than thin areas. If a thick region or feature exists in the shape of your part, it will shrink more than the thinner sections, which in turn will distort the shape of your thinner sections. The effect will be ellipses and warp.

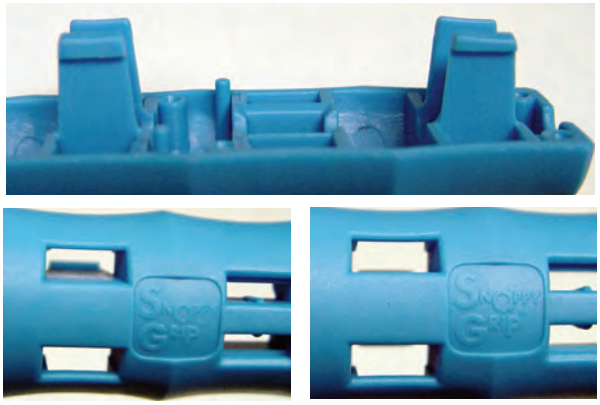
Orientation: The shape of a plastic molecule is long and chain-like. When in the relaxed state, these molecules are all jumbled together in every direction. But when you force molten plastic to flow through runner channels, gates, and through the cavity of your mold, you force those molecules to line up in the same direction. This is called molecular orientation. To some extent this orientation will relax – becoming once again jumbled – as the plastic cools. But some of the plastic freezes solid while still in this oriented state and remains oriented in the finished product. This affects the shape of your product because as the plastic cools, regions of oriented plastic shrink differently than regions that are in a relaxed state. These differences can contribute to surface defects and in some cases warp. When using filled materials, particularly fiberglass-filled materials, this phenomenon is magnified many times due to the shape of the glass fibers. Therefore, fiberglass-filled materials tend to be more challenging than non-filled materials to achieve roundness, flatness and straightness.

There are several things that can be done to maximize the roundness, flatness and straightness potential of a product, but with so many variables, these need to be determined on a case-by-case basis. The perfection of such features that are often taken for granted in machined metal products may be nearly impossible in plastics. When designing an injection-molded plastic product, consider the acceptable amount of warp and/or out-of-roundness while still achieving the design intent.

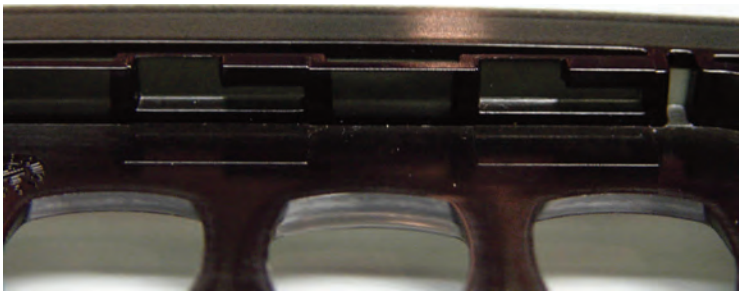
Tip #7: Recognizing and Avoiding Undercuts

An undercut is a design feature that prevents the plastic part from being ejected from the mold easily or properly. Some examples of undercuts include many "snap" features, holes in side walls, and threads. Experienced plastic part designers try to anticipate how the part will lay in the mold and where the parting line of the mold will be. They can then envision the two halves of the tool opening and the part ejecting from the tool. They can then anticipate design features that would obstruct the smooth opening of the mold or part ejection and attempt to avoid such features. It is wise to avoid these features if possible, because the tool design required to mold with undercuts will significantly increase the price of the mold, as well as the costs of maintaining the mold once it is built. The required cycle time to mold such a part may also be longer, resulting in increased piece price as well.

However, if a design requires an undercut, we can almost always provide that feature for you. We do it every day. But also be aware that such complexity does add cost. Minimizing undercuts can be a good way to help you maximize profits.



Notice that the top photo shows that this part has a "snap" feature with a hook on each of the tabs that extend from the back of the part. These typically require undercuts. However, the need for undercuts was eliminated by adding design features that opened the front of the part in direct alignment with the tabs (as illustrated by a slight rotation of the view of the part from the front). Thus, the mold created the tab without the need of moving slides (which account for the additional costs in mold design and cycle time).

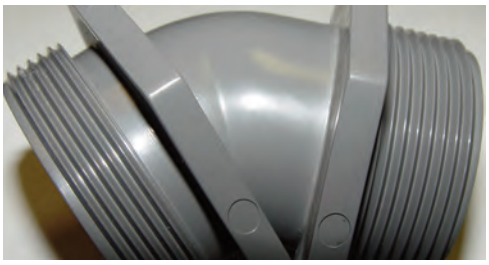


The undercuts were unable to be eliminated on this part due to the snap tabs located on the inside edge of this part. Slides were used.

Tip #8: Threads - Inside and Outside

Threads are a very popular design feature. Molded threads allow you fasten your plastic part to another plastic part, or to non-plastic products, by using common fasteners. In fact, those common fasteners themselves are often molded from plastic. Injection molding thread features is a well-established technology. However, there are some basic facts about producing them that are beneficial to know.

First, understand that threads represent undercuts. Both inside and outside threads are essentially undercuts, because they both inhibit ejection from the mold. Any time the design includes a thread feature, the cost of the mold will increase. Outside threads are often easier to create than inside threads. Outside threads can usually be formed by slides, mechanisms that actuate sideways when the mold opens, relieving the undercut situation.



Inside threads are more challenging. Inside threads require either a) an unthreading mechanism that spins the threaded core free from the molded part after the mold opens, or b) collapsible cores. As the name implies, collapsible cores collapse, they get smaller during ejection, thus relieving the undercut situation.

Slides, unthreading mechanisms, and collapsible cores all drive up the cost of building your mold, and most increase cycle time. If a design requires molded threads, no problem. We do that frequently. But keep in mind the costs during design, there may be alternatives.

Tip #9: Consider Cumulative Cost for Project

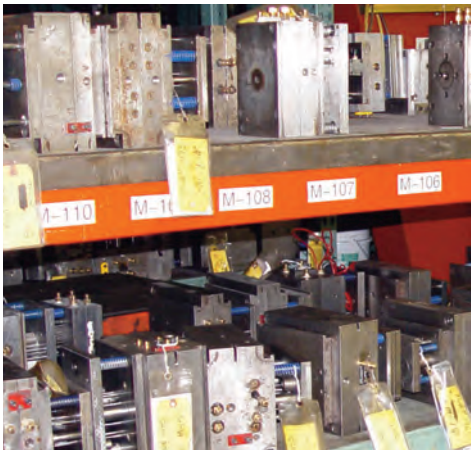
When designing injection-molded plastic products, there are two costs to be considered. The first is the per-piece cost to produce each part. This is the amount of money paid to the injection molder for each piece produced.

The piece price is affected by the cost of material, the amount of material used, and is also impacted by the cycle time (because machine costs and overhead costs are typically applied on a machine hourly basis). As an example, if the cycle time is 30 seconds, this equates to 120 parts per hour. If the hourly rate for the press being used is $24 per hour, this equates to an additional 20 cents per part for the piece cost.

Finally, the amount of labor needed to produce the product affects the piece price as well. This labor typically includes press-side personnel for any de-gating operation and/or inspection and finishing work that is required before shipment. Therefore, the less expensive the material selected, the less material used, the faster the cycle, and the less labor required, the lower the piece price will be.

The second type of cost is the tooling cost to build the mold. The cost of building a tool varies tremendously from one part to the next. Factors that affect tool cost include the complexity of the part design, the number of cavities built into the tool, the type and location of runners and gates, and the desired longevity of the tool. Long lasting tools – made with a higher grade of steel – cost more to build.

There is often a trade-off between the cost per piece and the tooling cost. For instance, building a four-cavity tool will almost always reduce the piece price because you get four pieces every time the machine cycles, versus one piece in a single-cavity tool. Building eight cavities would lower the piece price even further. Yet the cost of building the tool will go up significantly as well. For example, it would not make sense to spend an extra $20,000 on a multi-cavity mold in order to save $2,000 a year on parts for five years ($2,000 X 5 = $10,000).



Unfortunately, not all customers fully consider this trade-off relationship when finalizing their purchase decision. In many cases, the customer is focused heavily on piece price and ends up buying a more expensive tool than is necessary. Ultimately they are paying too much once the tool price is factored into the equation.

In other cases, the customer focuses heavily on getting an inexpensive tool and builds a single cavity tool to save money. But over the life of the product that customer may pay more for the parts and lose several times the savings realized from the low tooling investment.

Fortunately, Guttenberg Industries can make this decision clear and easy. Simply tell us how many parts you expect to need produced each year and how many years you expect the product to sell in the market, and we can recommend the number of cavities that will maximize your investment when we put your piece price together.

Tip #10: Materials and Tolerance

When designing a plastic part, material selection is critical. There are a lot of things to consider when choosing a material, such as strength, hardness, impact resistance, heat resistance, chemical resistance, and cost. The risk of selecting a material with unnecessarily high performance characteristics, needlessly driving up cost is nearly as great as selecting a material with too low performance for your application leading to field problems.

At Guttenberg Industries we have been molding with a wide range of materials for almost 40 years. We freely share our insights with our customers. The earlier in the process we become involved, the better our opportunity to help.

Selecting the best material for your product often involves a series of trade-offs between strength, impact resistance, flexibility, heat, UV resistance, and price. For instance, you may know that you need a strong material, but many of those break easily if impacted. You may want to know the best material alternatives that combine part strength with resistance to chemical attack. Perhaps you need to consider the least expensive material to meet your minimum design needs. To help in this decision process, we offer this Material Selection Comparison table that some designers find useful.

Material Selection Comparison Table

	1=Weaker 10=Stronger Tensile Strength	1=Weaker 10=Tough Impact Resistance	1=Soft 10=Hard Surface Hardness	1=Rigid 10+=Rubber Flexibility	1=Poor 10=Excellent Chemical Resistance	Max Contin. /Op. Temp	UV Resistance*	FDA/NSF Compliance & Notes	1=Less Expensive 10=Very Expensive Cost
Polypropylene	3	1 - 2	7	7 - 8	8	160	Fair	FDA UL	1
No-Break Polypropylene	2	No Break	4	7 - 8	8	160	Fair	FDA	1 - 2
Glass-filled Polypropylene	5	1	7	4 - 5	8	180	Fair		3
Talo-filled PP	3	1	5	5 - 6	8	180	Fair	USP Class IV, FDA-21	1 - 2
HD Polyethylene	3	1	4 - 5	7	10	180	Fair	FDA	1 - 2
LD Polyethylene	1		2	9	10	160	Fair		1 - 2
ABS	3 - 4	4	7 - 8	6	3	160	Poor	NSF	1 - 2
PC/ABS Blend	5 - 6	4 - 5	10	5	4	190	Fair		5 - 6
PC (Polycarbonate)	6	8	9	6	4	250	Poor	Food	4 - 5
Acetal (PCM)	3	1	9	5	8	180	Poor		3
Glass-filled Acetal	9 - 10	1	9	1	8	180	Poor		5 - 6
Acrylic	5 - 6	1	9 - 10	4 - 5	5	265	Good	FDA	4
Polystyrene	2	1 - 2	7	6	1	160	Fair	FDA	1 - 2
SB	2 - 3		5	7	3	160	Fair	FDA	2
Nylon**	4 - 6	1 - 8	4 - 9	5 - 7	8	149 - 295	Poor to Fair		3 - 10
Glass-filled Nylon	10	1	10	3	8	220	Poor		8
PBT	5	1 - 2	9	6	7	275	Fair		5
Glass-filled PBT	10	1 - 2	9	1	7		Fair		5
Polyurethane Rigid	4	10	9	6	7		Fair	NSF	9
Polyurethane Soft	2	No Break	1	1			Fair		9
TPE	1	No Break	1	10+	6		Fair		5
SEBS	1	No Break	2	10+	5		Fair		6 - 7
Ether-Ester	4	No Break	1	9	6		Fair	FDA Mult Automotive	2
Poyether Irnide (Ultern)	9 - 10	1	10	4		340	Excellent	FDA	10

*UV Resistance is stated for the BASE RESIN. Improved UV performance is often possible through UV Stabilization Additives.
**Nylon is a classification of material that includes a wide range of grades with a wide range of properties. Please contact us to discuss the best grade of Nylon for your project.

Material Tolerance Guidelines

Tolerance guidelines are expressed as a base (minimum) tolerance plus added tolerance based on linear inch of a given product feature, and represent typical variation from part to part once molded. By means of example, a 2 inch feature on a prduct molded from ABS will likely consume a tolerance of +/- .005 inches (Base of +/- .003 plus .001 X 2 inches). The same feature on a part molded from Polypropylene might consume a tolerance of +/- .007 inches (Base of +/- .003 plus .002 X 2 inches).

We urge caution in the application of these guidelines. Part geometry can have a dramatic impact of dimensional stability. In some cases, the guidelines provided are insufficient to cover actual variation, but in many more cases, these same guidelines suggest far more variation than is actually seen in the parts. The only reliable way to assess dimensional stability is to mold and analyze the product. At Guttenberg Industries, every project involves trial samplings and statistical analysis that compares actual variation to your print requirements. This process insures that every critical dimension as identified as key to quality is realistic as documented on the print.

Tolerance Rules of Thumb by Material

Material Specified	Rule of Thumb Tolerances	Material Specified	Rule of Thumb Tolerances
ABS	(+/-) 0.003 base plus .001 per inch	HD Polyethylene	(+/-) 0.005 base plus .0025 per inch
PC/ABS Blend	(+/-) 0.002 base plus .001 per inch	LD Polyethylene	(+/-) 0.004 base plus .002 per inch
Acetal	(+/-) 0.002 base plus .0015 per inch	Polypropylene	(+/-) 0.003 base plus .002 per inch
Glass-filled Acetal	(+/-) 0.002 base plus .0015 per inch	No-Break Polypropylene	(+/-) 0.003 base plus .002 per inch
Acrylic	(+/-) 0.003 base plus .001 per inch	Glass-filled Polypropylene	(+/-) 0.003 base plus .002 per inch
Nylon	(+/-) 0.002 base plus .0015 per inch	Talc-filled Polypropylene	(+/-) 0.003 base plus .002 per inch
Glass-filled Nylon	(+/-) 0.002 base plus .0015 per inch	Polystyrene	(+/-) 0.002 base plus .001 per inch
PBT	(+/-) 0.002 base plus .002 per inch	PUR (Rigid Polyurethane)	(+/-) 0.002 base plus .0015 per inch
Glass-filled PBT	(+/-) 0.002 base plus .0015 per inch	SB (Styrolux/K-resin)	(+/-) 0.003 base plus .001 per inch
PC (Polycarbonate)	(+/-) 0.002 base plus .001 per inch	TPE Santoprene	(+/-) 0.003 base plus .002 per inch
		TPE SEBS	(+/-) 0.003 base plus .002 per inch

Bring Guttenberg Industries your Toughest Jobs

We'll Gladly Accept Your Easy Ones Too!

Designing injection molded plastic parts can be complex and confusing. The task of designing in any material is already quite complex, but with injection molded plastic that complexity is compounded further due to the nature of plastics and the molding process. Fortunately, we are here to help.

We at Guttenberg Industries have built a solid reputation within the injection molding industry for our ability to conquer the toughest jobs in plastic. Many of our clients come to us for the first time after becoming frustrated by their experiences with other injection molding companies. These clients bring us the problem parts and we figure out how to resolve their issues. We use state of the art analysis tools like MoldFlow simulation software, SolidWorks 3D modeling systems, and rapid prototyping. We continually monitor the state of the industry for new technologies, materials and molding methods. Our internal experts have decades of experience in their chosen disciplines, and connections to outside experts if needed. Most importantly, we tackle each problem with a multi-disciplined approach that brings experts together from the fields of tool design, resin selection, plastic processing, and part design so that we are able to identify the root problem and do the job right.

Some injection molding companies turn away from the tough jobs. Some will raise your price or endlessly give you poor quality. Still others will try to convince you that your expectations are unreasonable. Not us! We welcome any challenge and boldly ask you to bring us your toughest jobs!

But remember that the best way to solve a problem is to prevent it from happening in the first place. Let us solve your tough job now, but next time let us prevent it from ever happening. Let our knowledge and expertise work for you.

For much more information on the capabilities of Guttenberg Industries, visit GuttenberInd.com.

You may also contact us directly at:

Guttenberg Industries, Inc.
603 S. Lincoln St., P.O. Box 70, Garnavillo, Iowa 52049
www.GuttenbergIndustries.com * (563) 964-1000
Greg Yoko GYoko@GuttenbergInd.com