How to Manufacture
A PERFECT PLASTIC PART

Four key factors that impact part quality

About This eBook

This eBook is dedicated to all of the product designers and engineers who are tasked with creating perfect plastic parts. We have outlined four key factors that go into making a high-quality plastic injection molded part: part design, tool building, material selection and manufacturing.

Each section contains the most important aspects of each stage in the manufacturing cycle along with some helpful charts and diagrams.

Table of Contents

Factor 1 – Part Design

Factor 2 – Tool Design and Build

Factor 3 – Material Selection

Factor 4 – Part Manufacturing

Conclusion

Factor One

Part Design

The process of plastic injection molding is designed to produce precision parts at a low cost. The part design must be developed to maximize the efficiency inherent in high-volume molding. With the right design, parts can be made consistently and with quality. Without a good design, costly processing mistakes can be made.

Around the industry, mostly everyone agrees that there are key design elements that must be held to a high standard for the injection process to work properly. They include wall thickness (t), rib design, boss design, corner transitions and weld lines and gate placement, and properly placed vents.

Part Design

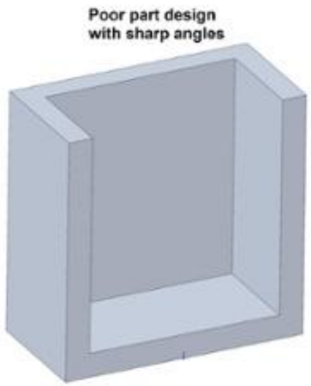
Designing for manufacturability

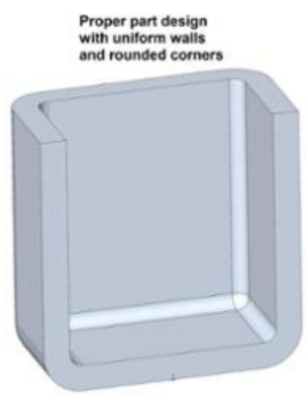
Wall thickness

Maintaining a uniform wall thickness in your part design will resolve most of the defects that can occur during manufacturing. Plastic, when melted, will flow into areas of least resistance. If you have a part with both thick and thin-walled sections, depending on the gate placement, the melt will flow into the thick walled areas first. The thin areas may not fill and pack properly. In addition, thicker areas will tend to cool more slowly and may be subject to voids or sinking defects.

If your part design does not have a consistent wall thickness, there are some things you can do to avoid improper fill. You can design the cavity, so the thicker areas transition gradually into the thinner areas.

The gate placement can also help ensure the cavity fills properly. However, it is still best to have a uniform wall. Uniform wall thickness helps the mold fill and cool properly. Thicker areas in the part design can be cored or hollowed out to maintain uniformity. These modifications will improve part quality and appearance.

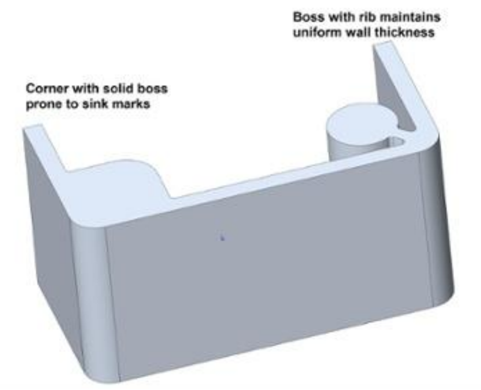




Part Design
Designing for manufacturability

Rib design

Ribs are used to help reinforce the overall strength of a part. Like flying buttresses, they support the walls and other dimensional components of a design. Depending on the material used, they should be between 50% to 70% of the wall thickness. Greater width could cause sinking to occur. To avoid this problem, a designer will often core out some material to reduce the shrinking. In addition, ribs cannot be too tall or too thin. The height recommendations are generally no more than 3 x the wall thickness. The corners should include radii and the height should include a draft (.5 to 1.5 degrees). The draft angle allows the part to be ejected from the mold.



Boss design

A boss is included in a part design to accommodate part assembly through screws or pins. They should conform to the same thickness as ribs. Thicker bosses will create sink defects as the part cools. As an alternative to a thick wall on a boss, the designer can use ribs to support the boss cylinder. And as with ribs, the boss also needs to have some draft to aid in ejection.

Part Design
Designing for manufacturability

Corner transitions

As we mentioned earlier, plastic melt flows into the area of least resistance. Sharp corners or angle changes impede this flow. These abrupt geometries can cause the cavity to not fill or pack properly, creating a part with defects.

Smooth transitions are all important in the injection molding process. So corners should have a curve versus an angle. The radius should be consistent on the inside and outside of the wall creating a uniform thickness. By using this design, the material will be able to flow throughout the cavity.

Weld lines

All plastic injection molded parts have weld lines. They key is to design the tool so that they do not compromise the part integrity. The part and tool can be built with varying geometries, wall thickness and strategic gating to place the weld in a position that does not interfere with the part performance.

Welds are formed when two material flows come together. This juncture often has cosmetic and structural problems. In fact, weld line considerations are one of the top issues facing injection molders. Computer aided flow simulators such as Solidworks Flow Simulation can help designers determine where the weld line will occur.

Part Design
Designing for manufacturability

Gate placement

The gates are the key to ensuring your mold fills properly. They direct the plastic flow from the runners for distribution throughout the part. Choosing the type of gate and its placement and will greatly impact the part quality and integrity. The wall thickness and the geometry dictate the size and placement of the gate. And, their location should minimize the flow length to avoid flow marks. Gates that flow from thick to thin walls will fill and pack better than the reverse. In some instances, additional gates may be needed to avoid cooling problems and shrinkage.

Vent placement

Vents will allow the gas (air) in the mold cavity to escape without causing super-heating or burning of the resin. They will also assist in minimizing weld lines if put in the proper locations.

Common Plastic Part Defects

Color/Additive Defects	Processing Defects	Mold Design/Maintenance
- Color Streaks	- Blistering	- Drag Marks
- Delamination	- Burn Marks	- Flash or Burrs
- Splay Marks	- Sink Marks	- Jetting
- Discoloration	- Flow Marks	- Warping

Custom plastic vials and tubes
Plastics range from opaque to transparent. Color additives can match specifications.

"Without a good design, costly processing mistakes can be made."

Factor Two

Tool Design and Build

A perfect, precision part begins with the mold. Building the tool takes time and a great deal of accuracy. It can also represent the largest investment in the manufacturing process, so getting it right is critical to the success of a project. If your goal is to manufacture parts with a high degree of precision in large-volume, the tooling becomes even more complex.

Tool Design and Build

When plastic meets the mold

The tool and the molding process are customized based on the type of plastic. Plastics that are amorphous are less free-flowing and tend to shrink less than crystalline or semi-crystalline plastics, which offer better flow, but higher shrinkage. For this reason, many projects call for engineering resins that offer a better melt and less shrinkage. Plastic suppliers provide information on the shrinkage rate of their resins along with temperature and melt flow rate recommendations.

Machining to exact standards

Resin shrinkage impacts the design and machining of the tool cavities. The cavities must accommodate for the amount of shrinkage that can occur. Using modern CAD software, the design engineer will create cavities that are larger than the actual finished part. Some of this shrinkage can be addressed by regulating the packing and holding rate in the mold, but all plastic shrinks as it cools, even after the part is ejected from the mold. Worse case, warpage can occur when a part has molded-in stress. This stress can be a result of issues with pressure, temperature, flow rate, gate location or venting.

Tool Design and Build

A strong mold design leads to high-volume quality

Precision parts can only be accomplished by meeting exacting standards not only in the cavities, but in the design of the mold components. Gates must be properly placed to allow for proper melt flow and pressure. The appearance of the final part can be improved by positioning the gates in an inconspicuous location on the part. The size of the gate is also an important consideration. The gate must be large enough to provide for proper packing of the material without extending the cycle time. If the gate is too small the packing may be insufficient to fill the cavity (also called a short shot) or the part may display other defects.

The design of the mold must also include vents. Vents allow the air that is displaced by the melted resin to escape the tool. As with gating, the size and position of the vents are key factors in producing a quality part. Vents that are too large can allow the plastic material to escape and cause flashing. Vents that are too small may not release enough of the trapped air and gas. These gas bubbles can cause an improper fill (short shot) or worse. The gas could combust and cause burn marks on the part.

Tool Design and Build

Proper cooling is key

An efficient and effective cooling system is the hallmark of a quality injection mold. The mold needs to maintain a consistent temperature to avoid shrinkage and warping while minimizing the cycle times to maximize production output. This delicate balance is achieved with a well-designed cooling system.

Ejecting parts

The final step in the molding process is releasing or ejecting the parts from the mold. The part geometry, type of resin, and mold finish are all considered when designing the ejector system. The placement of the ejector pins, the type of ejection mechanism and the cycle times need to be calculated with precision to avoid any defects in the part. This is generally accomplished with a series of carefully placed ejector pins, the size and position of which are determined by the shape, size and wall thickness of the part.

Running samples improves quality

Avoiding any unnecessary rework of a mold cavity will save time and money in the long run. Experienced molders create a sample mold that is used to produce a test run of the part. This step is vital in determining if any adjustments to the mold, the resin selection, or molding parameters such as temperature and flow rate are needed. If there are any quality issues, the project team will work together to determine the cause and re-sample the parts until they meet the customer's and the molder's standards.

Tool Design and Build

A quality precision mold, built to last for years is an investment in your company. Below we review the variables that impact the cost of a plastic injection mold.

The core metal

For shorter production runs, some mold makers will use molds made from aluminum. Aluminum is a perfectly reasonable choice if you will not need the mold to perform long-term. However, if a project requires that a mold lasts for several years, an aluminum mold may cost more in the long-run. High-volume precision molds are made from hardened steel. Steel can withstand the pressures of a long production run while still meeting close tolerances.

The number of cavities

It is pretty intuitive when you think about it. Fewer cavities in a mold require less tooling work, time and ultimately less cost. A reputable, experienced molder will be able to maximize cavitations in the mold to maintain the highest level of productivity. In general, most molders recommend creating one mold per part versus creating a family mold. Family molds are created with various cavities for assorted parts. They tend to produce inferior products and have more downtime due to maintenance issues.

The mold base

Think of the mold base as a case that holds all of the mold cavities, inserts and components together. The cost of the base is estimated based on the size of the mold and the type of steel used to make the base as well as the customization required.

Tool Design and Build

Core/cavity machining

All molds must also be customized. Customization includes the placement of cores, cavities, ejectors, cooling lines, etc. The steel used in the tool also impacts cost. Hardened steel molds last the longest and are more expensive to machine. Once done, however, they have a long production life.

Part complexity

Just as the number of cavities plays a role in determining the cost of the mold, so does part complexity. This includes the surface finish of the final part as well as the number of undercuts required. Parts, which demand tight tolerances, also contribute to the mold complexity.

Turnkey or vertically integrated injection molders

Some mold builders also manufacture the parts. This type of integration can help defray the mold building cost. Often full-service molding manufacturers will subsidize a portion or all of the cost of the mold based on the full term and value of the manufacturing contract. They will amortize the cost of the mold so they can maintain profit margins while providing the lowest possible per piece cost to their clients.

"Building a tool is a large investment in time and money. Getting it right is critical."

Factor Three

Material Selection

There is a science to choosing the right plastic resin material for a part. This section focuses on the basic characteristics of polymers with some examples of popular resins and their applications.

Material Selection

To select the best resin for your manufacturing project, answering the questions below will help guide you to the right material.

First, what is the intended end use of the part?

- Does the part need to be rigid or flexible?
- Does the part need to withstand pressure or weight?
- Will the parts need to withstand certain temperature variations?
- Will the parts be exposed to other elements or chemicals?

Second, are there special appearance considerations?

- Is a specific finish required?
- Does a color need to be matched?
- Is embossing a consideration?

Third, what, if any, regulatory requirements apply?

- Will the product be exported and need to meet REACH standards?
- Does the part need to be food safe?
- Will children use the product?

Determining the right resin for an application, is key to part quality and performance.

Material Selection

A plastics primer – thermoset vs. thermoplastic

Plastics fall into two basic categories, thermoset plastics, and thermoplastics. To help you remember the difference, think of thermosets just as the term implies, they are "set" during processing. When these plastics are cured, it creates a chemical reaction that "sets" the part into a permanent form. The chemical reaction is not reversible, so parts made with thermosets can't be re-melted or reshaped. These materials can be a recycling challenge unless a bio-based polymer is used.

Thermoplastics are heated then cooled in a mold to form a part. Once the parts are cooled, they revert to their original state and can be re-melted and cooled again. For this reason, thermoplastics are easier to reuse and recycle. They comprise the majority of the manufactured polymer resins on the market today and are used in the injection molding process.

Thermoplastics	vs. Thermosetting plastics
- Can melt when heated	- Shaped through chemical reaction
- Can be recycled	- Cannot be recycled or reshaped
- Can be reshaped and molded	- Heat resistant
- Chemical resistant	- Dimensional stability

Material Selection

Fine tuning the resin selection

Thermoplastics are categorized by family and type. They fall into three main categories or families; commodity resins, engineering resins, and specialty or high-performance resins. The high-performance resins also come with a higher cost. That is why commodity resins are often used for many everyday applications. Commodity resins are easy to process and are inexpensive. They are often found in common mass produced items like packaging. Engineering resins are more expensive but offer better strength and resistance to chemicals and environmental exposure.

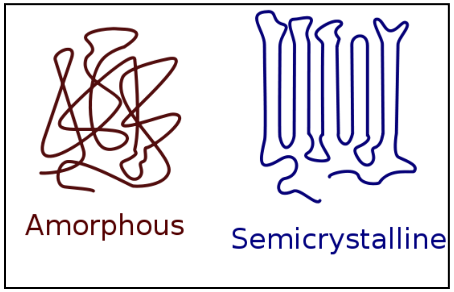
Within each of these families, there are resins that have different morphology. Morphology describes the arrangement of molecules in a resin and fall into two categories, amorphous and semi-crystalline. Amorphous resins have the following characteristics: they shrink less when cooled, offer better transparency, work well for tight-tolerance applications, tend to be brittle and lack chemical resistance. Semi-crystalline resins have the following characteristics: tend to be opaque, offer good abrasion and chemical resistance, are less brittle and have higher shrinkage rates.

Material Selection

Examples of available resins

Amorphous

An example of an amorphous, commodity resin is polystyrene or PS. Like most amorphous resins, it is transparent and brittle, but it can be used in high-precision applications. It is one of the most widely used resins and can be found in plastic cutlery, foam cups, and plates.



Wiki commons attribution: Polymerketten-amorph_und_kristallin.svg: Rainer Ziel.

Higher up on the amorphous scale are the engineering resins such as polycarbonate or PC. It is temperature and flame resistant and has electrical insulating properties, so it is often used in electronic components.

An example of a specialty or high-performance amorphous resin is polyetherimide or (PEI). Like most amorphous resins, it offers strength and heat resistance. However, unlike most other amorphous materials it is also chemically resistant, thus often found in the aerospace industry.

Material Selection

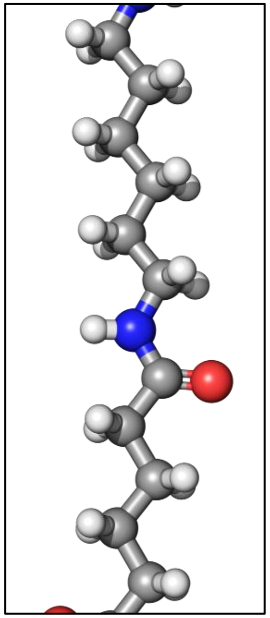
Examples of available resins

Semi-crystalline

An inexpensive semi-crystalline commodity resin is polypropylene or PP. As with most semi-crystalline polymers, it is flexible and chemically resistant. The low cost makes this resin the choice for many applications such as bottles, packaging, and pipes.

A popular engineering, semi-crystalline resin is polyamide (PA or nylon). PA offers chemical and abrasion resistance as well as low shrinkage and warp. There are bio-based versions available making this material an earth-friendly alternative. The toughness of the material makes it a light-weight alternative to metal in automotive applications.

PEEK or polyaryletherketone is one of the most widely used semi-crystalline high-performance resins. This resin offers strength as well as heat and chemical resistance and is often used in demanding environments including bearings, pumps, and medical implants.



Wiki commons attribution:
This 3d image of Nylon was created with PyMOL by Yassine Mrabet

Material Selection

Amorphous Resins	Semicrystalline Resins	Cost
High-performance/ Specialty ***Polyetherimide or PEI*** High strength & heat resistance Chemical resistant Used in aerospace applications	High-performance/ Specialty ***Polyetheretherketone (PEEK)*** Heat & chemical resistant Strong and corrosion resistant Used in bearings, medical implants	EXPENSIVE
Engineering ***Polycarbonate or PC*** Transparent Heat & flame resistant Electric insulator Used in electrical components	Engineering ***Polymide or PA (Nylon)*** Chemical & abrasion resistant Low shrinkage & warp Used in auto parts, carpet and clothing	MODERATE
Commodity ***Polystyrene or PS*** Transparent Low strength Low heat resistance Used in plastic cutlery, Foam plates and cups	Commodity ***Polypropylene or PP*** Flexible & tough Chemical & fatigue resistant Used in bottles, packaging & living hinges	INEXPENSIVE

Material Selection

Plastic additives to improve characteristics

Various resins have particular properties for which they are known. As we have seen, resin families (commodity, engineering, and high-performance/specialty) contain both amorphous and semi-crystalline alternatives. However, the higher the performance, the higher the cost. To help keep costs low, many manufacturers use additives or fillers to obtain the qualities they need at a lower cost. These additives can be used to improve performance or convey other characteristics to the final product.

Below are some of the most common additive applications:

Antimicrobial – Additives used in food-related applications or high-contact consumer products.

Antistatics – Additives used to decrease the static electricity conduction and are often used in sensitive electronics.

Plasticizers and fibers – Plasticizers make a resin more pliable, whereas fibers add strength and stiffness.

Flame retardants – Additives used to make products resistant to combustion.

Optical brighteners – Additives used to improve whiteness

Colorants - Additives that add color or special effects such as fluorescence or pearlescence.

Material Selection

The final selection

Choosing the right material for a project is one of the most important factors in creating perfect plastic parts. The advances in polymer science have contributed to the development of a large selection of resins from which to choose. It is important to work with an injection molder that has experience with a variety of resins and applications including resins that are compliant with FDA, RoHS, REACH and NSF.

Over the years, The Rodon Group has developed strategic relationships with the best resin suppliers in the country. We have over 50 years of experience using certified commodity and engineering resins that adhere to our stringent manufacturing standards. Unless you are well-versed in resin selection, you should always consult with your injection molder to determine the best material for a particular project.

Sources: American Chemistry Council; Georgia Institute of Technology, Atlanta, GA; RTP Engineering Plastics; Blackwell Plastics; British Plastics Federation; Hardie Polymers; Wikipedia

Factor Four

Manufacturing

The final factor in creating a plastic injection part is the manufacturing phase

Manufacturing

So far we have reviewed the first three factors that contribute to a successful plastic injection project including part design, tool design and build, and material (resin) selection.

To recap; the part has been designed for manufacturability, the resin has been selected based on the requirements of the part, and the mold has been built to maximize speed without forsaking quality.

Time to get the presses rolling? Not quite yet.

Before a production mold is ramped-up and running, a sample mold is created to test the cavitation and the resin. This vital step can help avoid costly reworks of an injection mold. If any problems surface, they can be addressed before the production mold goes into service.

Manufacturing

Choosing the right injection machine for the job

Injection machines (presses) come in all shapes, sizes and configurations. The selection of the machine will depend on several variables including the size of the mold, the number of cavities and the selected resin. Most plastic injection companies will provide a molding equipment list on their website. It may look something like this:

3- 68 Ton Injection Molding Presses
5- 123 Ton Injection Molding Presses
5- 154 Ton Injection Molding Presses
5- 202 Ton Injection Molding Presses
5- 233 Ton Injection Molding Presses
4- 400 Ton Injection Molding Presses

So, what does this mean? Plastic injection molding machines are classified or rated based on tonnage, or more specifically the clamping pressure or force. Presses can run in size from less than 5 tons of clamping pressure to over 4000. The higher the press ton rating, the larger the machine.

A machine rated for 68 tons can deliver 68 tons of clamping pressure. This pressure keeps the mold closed during the injection process. Too much or too little pressure can cause quality issues. Too much or too little pressure can also cause flashing, where the excess material appears on the part edge. Pressure also impacts the viscosity of the plastic being used in the project. Melt Flow Index or MFI is a measure of the ease of flow of the melt of a thermoplastic polymer. Plastic compounds react differently to pressure based on their MFI. The higher the MFI, the higher the pressure needed.

Manufacturing

Choosing the right injection machine for the job, continued

How much clamping force or pressure is required? There are many factors that are taken into consideration when determining the size of the press. The size of the part, the polymer being used and something called the safety factor. The safety factor is an additional numerical percentage buffer that is added to the calculation to help avoid defects in the final part. Some recommend adding 10% to allow for the safety factor. As mentioned earlier, the MFI (Melt Flow Index) of the plastic compound will also impact the pressure needed to produce the part. Many calculations include the platen size as well as the mold and part size; however, to get an estimate of the press size your project will need, we have simplified it even further.

Many plastic injection professionals use a general rule of thumb of 2.5 times the surface square inches of the part to be produced. So, if you have a part with 42 square inches then you would need a press size with 105 tons of pressure. If you add 10% for a safety factor, you will need to use a press with a minimum of 115 tons of clamping force. A press size of 120 tons would be able to accommodate your plastic injection molded product.

Manufacturing

Roll the presses

When the production mold is finalized, and the machine has been prepped, manufacturing begins. An initial run will be thoroughly examined for any part defects. If none are found, full production can begin. Depending on the part and the agreement with the molder, quality checks can take place on a regular basis. Most reputable molders will do their own quality checks. They will check for strength, color correctness, and any common defects that can occur like flash or warping.

Automation = Efficiency

Now it comes down to inventory. You need to be sure you have the right quantities in the right place at the right time. At Rodon, we keep a safety stock for our clients to ensure we can meet any unseen demands. We also monitor the quantities on hand, in real-time, so we can help our clients estimate their future needs. This saves everyone some heartache.

We utilize an MRP system (Material Requirement Planning system).This tool allows the manufacturing team at Rodon to monitor jobs from the minute they are initiated to the time they are delivered including all of the downstream impacts. According to Plant Manager, Tom Moore "The team manages to our customers requirements. By monitoring production runs throughout the day, we can ensure the needs of the customers are met. We manage the efficiency of each job. These measures ensure we always have the materials and products required."

Our demand-driven planning allows us to meet the JIT manufacturing needs of our discerning clients. Rodon's manufacturing team's experience with cycle times, resin properties, and part demand has resulted in a 99.8% satisfaction rating. When you are looking for an injection molder, ask them how they monitor and manage workflows and processes. Professional companies should have manufacturing systems in place that help improve efficiencies and lower costs.

Manufacturing

Got questions? We have answers.
We get a lot of questions at Rodon about the plastic injection process and how we can meet the needs of our clients. Below we have listed some of the most frequently asked inquiries. If you would like to get additional information, please visit our Contact Us page and we'll try our best to get you the answers you're looking for.

What are the minimum quantities you can order? Rodon is a high-volume plastic injection molder. Our molds are built to exact specifications and can withstand years of continuous use. Given the resources we invest in making the highest quality molds, we begin our production runs with a minimum order of 1,000,000 parts or more.

Do you make prototype tools? We make prototypes based on the needs of the customer, the application and the size of the part. We do not offer "job shop" molding and prototyping. When we make a tool, we invest a great deal of time ensuring the design is precise while being cost effective. We offer financing terms and other payment arrangements to help defer the cost of the mold.

Can you provide a quote on making a plastic injection mold? Yes, we provide the tool making cost when we supply a quote to produce a part. To accurately quote a part we need to get answers to a few questions. You can fill out our quick quote form to start the process.

Manufacturing

Got questions? We have answers, continued…

Does your company sign Non-Disclosure Agreements (NDAs)? Yes, as an American manufacturer, we are all too familiar with the problems some of our customers have had with foreign producers, who may or may not honor NDAs. We review the NDAs with in-house counsel to ensure the intellectual property of our customers is fully protected.

How long does it take to produce a part? The length of time needed to get your parts delivered is determined by several factors. First, the tool needs to be made. Depending on the complexity, this stage will take approximately three months. Once the tool is made, we do a short production run to ensure the customer is satisfied with the final part. If adjustments are required, additional time may be needed. With the tool and sample parts finalized, the production can begin.

Do you offer extended services such as assembly, packaging or decorating? We offer some packaging at the press but do not offer custom packaging, decorating or assembly at the press. Additional secondary operations can be performed by approved 3rd party vendors with whom we work.

You say you beat China pricing, how do you do that? We have invested a great deal in new equipment and robotic automation. We pride ourselves in producing the highest-quality tools available ensuring hassle-free production for years to come. These factors allow us to provide low-cost per piece pricing right here in America. Globally competitive prices and elimination of offshore risks add up to a delivered part price that makes us the envy of our industry. See our Cheaper that China comparison chart to see how "Rodon Beats China Pricing."

Conclusion

Choose an experienced injection molder for your next project

Manufacturing a perfect plastic part takes years of dedication and experience. You need a combination of talented engineers, tool builders and manufacturing staff to produce consistent quality parts over time. The Rodon Group has been providing a premier turnkey manufacturing solution to hundreds of clients across many industries. Find out how we can help you.

Contact us today for a project review