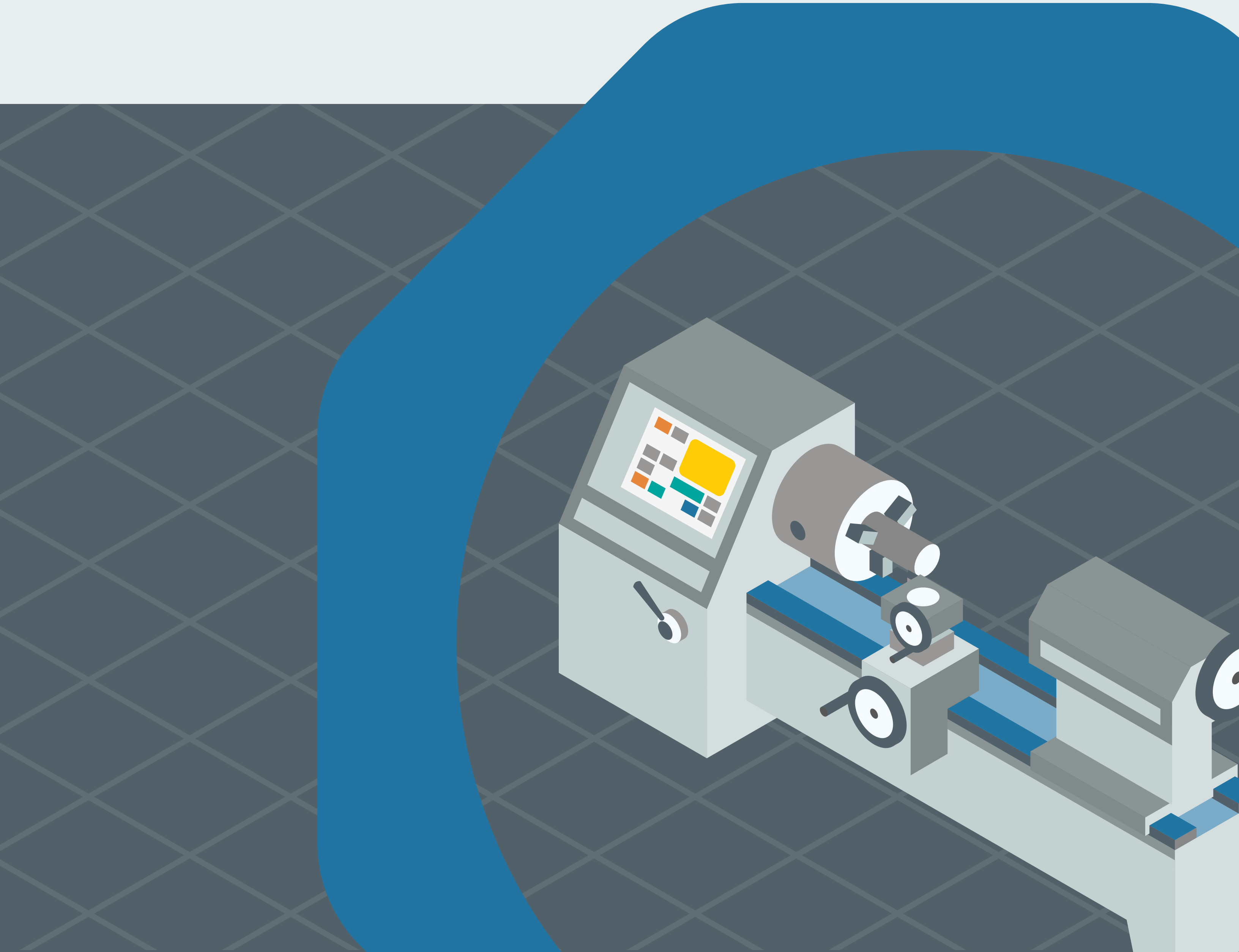
An Introduction to Plastic Injection Molding

THE RODON GROUP®



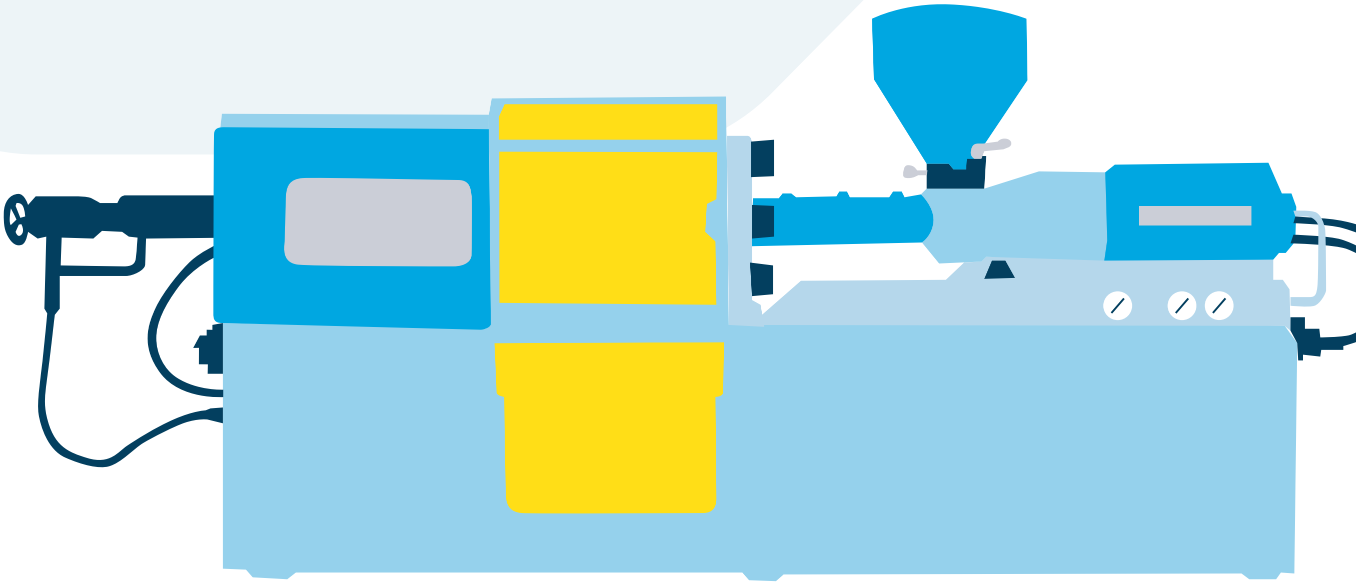
A resource to help designers, engineers and purchasing professionals navigate the world of plastic injection molding

Table of Contents

Introduction

We developed this eBook with designers, engineers and purchasing specialists in mind. It is written to provide a basic understanding of plastic injection molding presses, processes and costs. Our goal is to make our customers more knowledgeable about what goes into making a plastic part. We hope you find this eBook informative and useful. Please feel free to share it with your colleagues.

Understanding
Plastic Injection Molding



In the plastic injection molding process, a stainless-steel part mold is filled with molten plastic, which then cools to form the final part. This manufacturing process offers a cost-effective method for producing large volumes of parts with consistent dimensions. Production volumes for plastic injection molding projects often range into hundreds of thousands or millions of parts.

While the initial tooling is more cost-prohibitive than many machining methods, the economies of scale gained in large production runs negate the initial set-up costs. The process also produces very little scrap and wasted thermoplastic material can be recycled back into the machine for use in production.

The Rodon Group has been at the forefront of thermoplastics manufacturing since its inception in 1956. We have 118 injection molding machines in our Pennsylvania facility, making us one of the largest family-owned and operated injection molders in America.

Injection Molding

Plastic injection molding remains the most versatile of plastic molding processes. A comprehensive range of injection molding machinery exists in differing tonnages or pressure ratings, and common material choices include multiple plastic resins and a broad range of additives used to alter the physical properties of final parts. Based on machine type and material choice, injection molding can produce items ranging from large automobile parts to small precision medical components.

While fabricating steel molds for plastic injection molding is expensive, this cost is offset by the low cost-per-part offered by the process. Low part cost, design flexibility, a variety of material and finishing options, and expansive customization techniques have made injection molding one of the most popular manufacturing processes available for engineers.

The standard injection molding manufacturing process follows these steps:

1. Plastic is heated to its melting point.
2. The melted plastic is injected into the mold with a nozzle until all cavities are filled.
3. The mold is cooled, creating a finished part.
4. The finished part is ejected from the mold automatically using pins.

Designing Plastic Injection Molding Parts

Developing a design for plastic injection molding is a complicated process that requires significant skill to avoid manufacturability issues. Proper design, planning, prototyping, and testing will help to prevent many of the pain points that can arise during the early phases of manufacturing. The right design will facilitate the production of high volumes of specialized, high-quality plastic parts without any costly delays or expensive defects.

Designers or design contractors should account for the following aspects when creating a design for a plastic injection molded part:

- Boss design
- Corner transitions
- Gate placement
- Rib design
- Vent placement
- Wall thickness
- Weld lines

Learn more about part design in the "Key Considerations for Achieving the Perfect Plastic Part" section of this eBook

Material Selection for Plastic Injection Molding

Selecting the appropriate resin for a design is critical for the ultimate functionality of the final part. Recent advancements in polymer science have produced a comprehensive range of plastic resins that can facilitate a wide variety of mechanical and aesthetic properties in molded parts.

There are five factors to consider when choosing a plastic resin:

1. The aesthetic appearance of the part (color, texture, transparency)
2. The strength, stress, and flexibility/rigidity requirements of the part
3. Required resistances (temperature, chemical, weather, etc.)
4. Part service life
5. Regulatory requirements (FDA, RoHS, REACH, NSF)

With so many materials available, resin sourcing can be confusing. Working with an experienced injection molding provider ensures you'll get the correct resin for your design.

Fabricating Molds for Plastic Injection Molding

Constructing the molds used for plastic injection molding is the most time-consuming and expensive part of the process. These tools require a great deal of precision and specialized consideration by talented design engineers and toolmakers. An error in the mold could result in thousands of parts being ruined. As such, quality control and careful testing before full production are critical to project success.

While mold-making is the most expensive step in the plastic injection molding process, a quality precision mold will last for many years and can potentially facilitate the creation of millions of products. When creating a mold, there are multiple variables to consider, including:

Core metal

Most high-volume molds are fabricated from stainless steel. The steel can be machined and fabricated to facilitate close tolerances and complex geometries, but still lasts for many years. At The Rodon Group, we still use many stainless steel molds that were created decades ago.

Required cavities

Mold cavitations should be maximized to support higher efficiency during production. Also, using one mold per part in a multi-cavity tool typically facilitates uptime and high precision, while family molds tend to require significant maintenance and produce inferior parts.

Mold base

A mold base is required to hold all mold cavities, components, and inserts in place during manufacturing. The cost of a mold base can vary wildly depending on customization requirements, size, and material.

Machining

A variety of machining capabilities will be required to implement necessary customizations into the mold, such as cavities, cooling lines, cores, and ejectors.

Complexity

More complex part designs will increase the ultimate cost of constructing a mold.

Learn more about molds in the "Key Considerations for Achieving the Perfect Plastic Part" section of this eBook

Plastic Injection Molding Costs

The start-up costs associated with plastic injection molding can vary wildly based on the complexity of the design. The injection mold itself is typically a significant investment—even more so if the design requires tight tolerances, intricate geometries, or if the mold has multiple cavities.

Other factors that may influence overall costs include:

- Resin type
- Part size
- Cycle times

Ultimately, the costs of plastic injection molding scale downward as volume scales upward. This is why plastic injection molding is considered a high-volume process—the set-up costs are often too prohibitive for small and medium production runs.

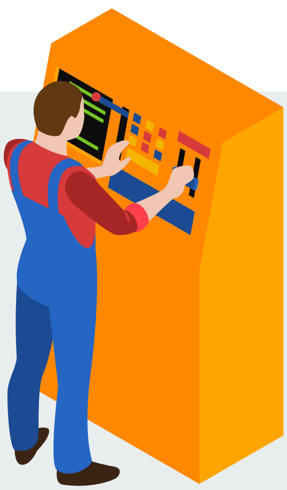
Learn more about costs in the "Determining the Cost of an Injection Mold" section of this eBook.

The Plastic Injection Molding Manufacturing Cycle

Once you've committed to plastic injection molding as your process of choice, you'll need to source a quality provider for the service. At The Rodon Group, our plastic injection molding process follows these steps:

Machine selection.

We'll work with you to select the appropriate machine size for your mold and material choice. These machines are classified based on their tonnage rating, which refers to the amount of force they apply when clamping. At The Rodon Group, we have machines with capacities from fractional to 107 ounces and clamping force from 46 to 720 tons.

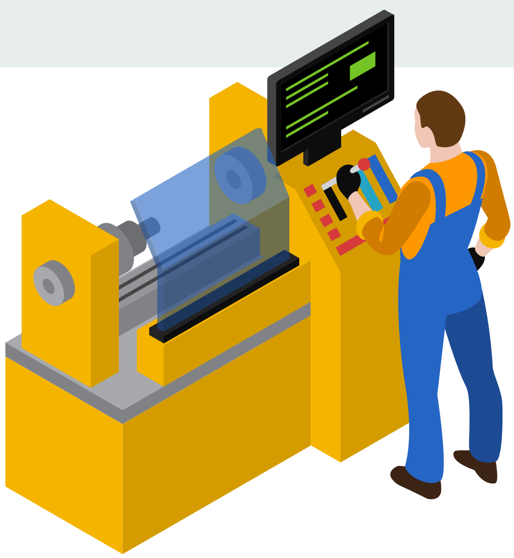


Preparation

Once the mold is ready, and the machine and resin material is selected, an initial run can begin.

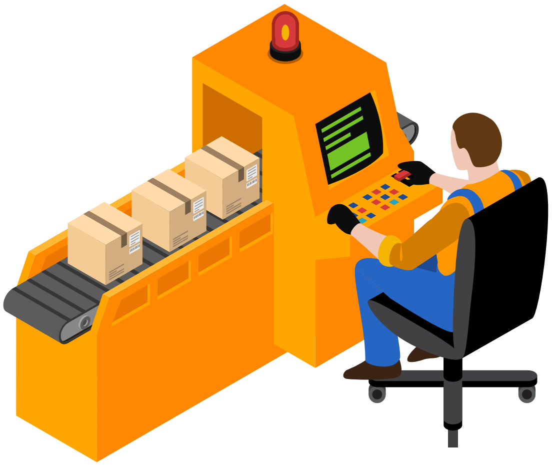
Initial Run

A small initial production run will undergo stringent quality assurance examinations to identify potential defects or other problems. Once the initial run has been cleared, full production begins.



Full production

Once full production is underway, quality checks will continue at regular intervals to ensure no problems arise. Quality assurance measures check for common defects such as warping or flashing, part strength, and coloring issues.



Inventory

Production planning based on your predictions and forecasts should ensure you have inventory when you need it. However, The Rodon Group understands that emergencies and unexpected sales peaks do happen. We keep safety stock on hand for every client to meet sudden demand spikes.

MRP

We also strictly monitor all material inventories in real-time using our Material Requirement Planning (MRP) system. This lets us support client inventory planning initiatives and monitor jobs from planning through delivery.

Cold Runner vs. Hot Runner Molds

Plastic injection molding uses two different mold types: cold runner and hot runner. Each type has its benefits and drawbacks. The experts at The Rodon Group will help you determine which method is best for your project.

Cold Runner Molds

The mold base of a cold runner mold typically has two or three plates held within it. Plastic is injected into the mold using a sprue. This plastic fills up the runners and travels into the mold cavity.

- In two-plate cold runner molds, the runner system is attached to the mold, and an ejection system is in place to separate them.
- In three-plate cold runner molds, the runner is separated on its own plate, so only the finished part will be ejected.

In both two- and three-plate cold runner systems, these runners can be reground and recycled. This process increases overall cycle times but also reduces plastic waste.

Advantages of cold runner molds include:

- Cost-effective
- Low maintenance costs
- Can use many different polymer resin types
- Engineered or commodity
- Facilitates fast color changes
- Improved cycle times when robotic assist is available to remove runners

Disadvantages of cold runner molds include:

- Slower cycle times compared to hot runner systems
- Increased waste if plastic runners aren't recycled

Hot Runner Molds

Hot runner molds are built from two plates heated by a manifold system. The manifold heats the plastic, and then the mold cavities are filled with molten plastic via nozzles. While a variety of hot runner systems exist, they can largely be split into two categories: internally heated and externally heated.

- Internally heated hot runner systems offer better control of plastic flow.
- Externally heated hot runner systems are best for polymers that are sensitive to changes in temperature.

Hot runner systems eliminate runners, so there is no need to recycle or regrind runner plastics. Insulated runners are variations on the hot runner system which are used for injecting semi-crystalline polymers with low levels of thermal conductivity. Instead of using heat to keep the plastic in its melted state, the insulation on the runners keeps it from cooling.

The advantages of hot runner systems include:

- Less waste
- Potential for faster cycle times
- No need for automation to remove runners
- Can handle large parts

The disadvantages of hot runner systems include:

- Molds are more expensive to create
- Difficult to change colors
- High maintenance costs
- Higher potential downtime
- Often not suitable for thermally sensitive resins

Learn more about molds in the “Cold Runners vs. Hot Runners” section of this eBook

Types of Plastic Molding

In today's ever-evolving world of manufacturing, plastics are being used to make everything from automotive body parts to synthetic human body parts. To create critical components and ensure optimal performance, many manufacturers choose to make use of plastic injection molding.

To suit diverse industry needs, there are several other types of plastic molding available, each offering unique features and benefits. An application's particular specifications and requirements will determine which type of molding process is best-suited for a specific part.

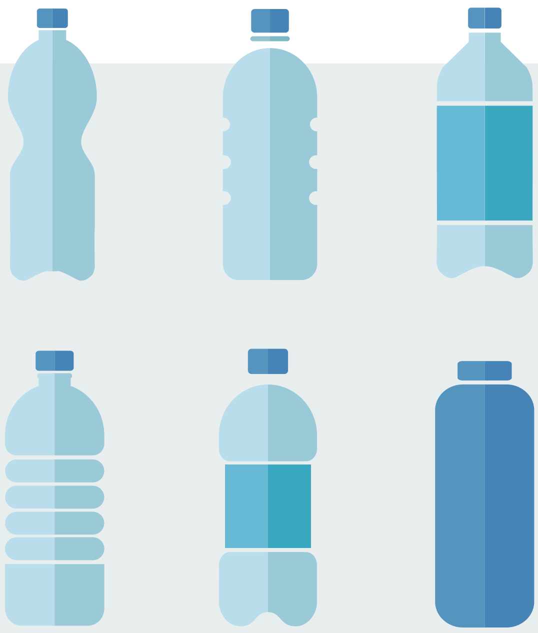
Below are a few of the most common different styles of plastic molding, including injection molding.

Blow Molding

The blow molding process follows the same basic steps found in the art of glass blowing. To blow mold a part, the manufacturer inflates a parison — a heated plastic mass, usually in the shape of a tube — with air. The parison inflates until it fills the mold and conforms to its shape. In this way, the plastic is blown into its desired form. Once cooled, the newly formed plastic part is ejected from the mold.

Blow molding is especially useful for economically manufacturing one-piece, hollow objects in large volumes, as the process can quickly create uniform, thin-walled containers — perfect for small objects like bottles, as well as larger ones like storage containers and drums.

Depending on the specific application, manufacturers can use a variety of thermoplastics in blow molding to create a more customized product. Commonly worked materials include low-density polyethylene, high-density polyethylene, polyethylene terephthalate, polypropylene, and polyvinyl chloride.



Blow molding is especially useful for economically manufacturing one-piece, hollow objects in large volumes, like bottles.

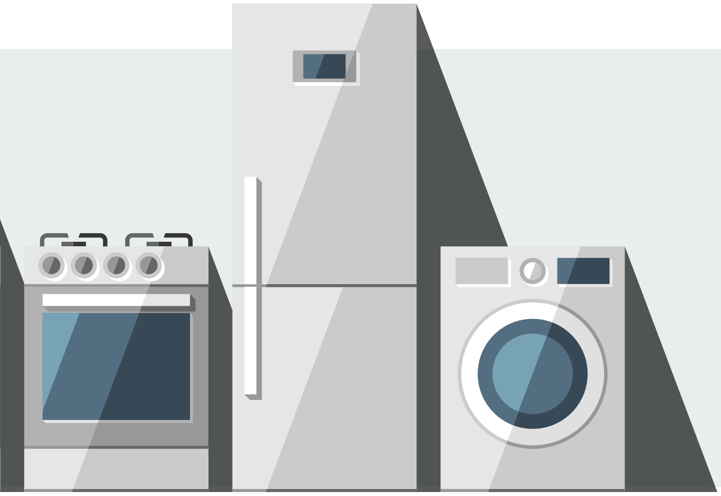
Compression Molding

Compression molding involves placing a heated plastic inside of a heated mold, then closing it to compress the plastic into the desired shape. Once cooled, the part is removed from the mold. The heating process, called curing, helps ensure that the final product will maintain its integrity and shape.

Compression molding offers many unique advantages; it's both cost-effective and highly efficient. The process is also quite versatile, allowing manufacturers to create parts that vary greatly in thickness, length, and intricacy.

Because compression molding often uses advanced composites for the plastic material, the process yields stronger, more durable parts, making it popular across a range of different industries. For example, compression molding often employs high-strength materials, such as thermosetting resins, fiberglass, and reinforced plastics, resulting in products that are sturdier and more resilient than those offered by other molding processes.

Allowing for the creation of high-strength parts, compression molding is used to produce components for a vast range of applications, including automotive parts, household appliances, clothing fasteners, and body armor.



Compression molding is used to produce components for a vast range of applications, including household appliances and more.

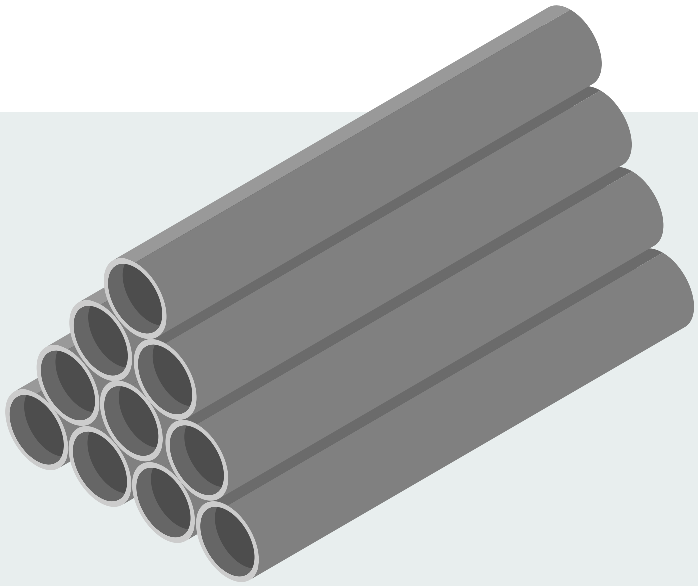
Extrusion Molding

While other forms of plastic molding use extrusion to insert the plastic resins into the mold, extrusion molding extrudes the melted plastic material directly into the die. This process is unique in that the shape of the die, not the mold, determines the shape of the final product.

Extrusion molding is ideal for manufacturing parts with continuous length and uniform cross-sections. Similar to a plastic injection molding machine, the extrusion molding machine has a screw that turns to feed the plastic resin into the feeder. The molten plastic then moves through a die, creating a long, tubular shape. The shape of the die determines the shape of the plastic tube. Once the extrusion is cooled, it is removed from the machine.

Extrusion molding is well-suited for long, hollow-formed applications, such as tubes, pipes, and straws. Plus, manufacturers can create these parts in many different shapes, including T-sections, U-sections, square sections, I-sections, L-sections, and circular sections.

Extrusion molding is well-suited for long, hollow-formed applications, such as tubes, pipes, and straws.

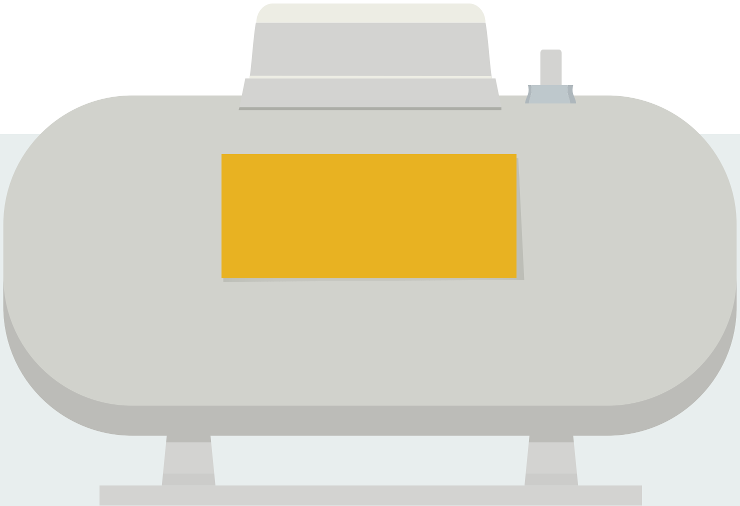


Rotational Molding

Rotational molding, also known as rotomolding, uses high temperatures and rotational movement to coat the inside of a mold to form a part's desired shape. First, the mold is filled with a polymer powder. The heated mold then rotates on two perpendicular axes so that the powder adheres to the entire interior of the mold. As it continues to rotate, the mold eventually cools and is removed, ultimately forming an even-walled component.

Rotational molding is best suited for the creation of large, hollow, one-piece containers, such as tanks. Though cost-effective, it is not a fast-moving process. However, rotomolding wastes little material, and what excess material is produced can often be reused, making it an economical and environmentally friendly manufacturing process.

Other key advantages include the ability to produce parts with consistent wall thicknesses, enhanced design flexibility, and great strength.



Rotational molding is best suited for the creation of large, hollow, one-piece containers, such as tanks.

Injection Molding

Of the various molding processes available, injection molding is considered to be the most versatile, as it can be used to create a variety of parts, ranging in both size and shape. Presses also come in different sizes, based on the pressure they exert and their tonnage. Injection molding basic principles are fairly simple, but the actual process can be quite complex when it comes to maintaining part consistency. The process involves the injection of melted plastic into a mold, which is made of steel. The mold itself has cavities that will form the parts; once injected, the molten plastic fills the cavities and the rest of the mold. Once cooled, the parts are ejected by pins.

Thanks to its excellent versatility, injection molding can be used to create everything from large automotive parts to small, intricate parts used in surgical equipment. Injection molding also allows for a high level of customization, as various plastic resins and additives can be used, allowing designers and engineers to create unique parts to meet highly complex or unusual application needs. And there are some enhancements and techniques available — such as an array of resin and finish options — for manufacturers looking to create even more specialized parts.

Though it can be expensive to initially make the molds themselves, once built, the production costs become quite low. In fact, injection molding is best-suited for the creation of very high volumes of precise parts; once production begins, the cost per part drops significantly, making the process very economical for high-volume runs.

Plastic injection molding is a highly reliable solution for producing large numbers of precise, consistent components. It's also more efficient and cost-effective than other molding styles, in that it produces much less waste. As a result, injection molding is most often used for the manufacture of high-quality parts in high volumes.

Key Considerations for Achieving The Perfect Plastic Part

There are several key elements that are essential for creating the perfect plastic part. At The Rodon Group, our dedicated team of plastic injection molding experts fully understands what it takes to create high-performing, reliable components, and, unlike most other companies, we pride ourselves on being a complete turnkey precision component manufacturer, designing and tooling all parts in-house and carefully overseeing every step of the production process.

Below, we share our key areas of focus in the plastic injection molding process; together, these critical elements allow us to manufacture the ideal plastic part for each of our customers, no matter how complex or unusual the specific application.

Mold Design

First, the design of the mold is critical for ensuring that the plastic injection molding process can produce a high volume of parts with optimal efficiency and consistency. At Rodon, our team of engineers excels in the design of molds that deliver high-quality parts the first time around — eliminating the risk of costly delays and material waste.

With more than 60 plus years of experience in the field, our engineers have the skills and creativity needed to create any type of custom mold, even for the most challenging situations. Using state-of-the-art technology, including CAD/CAM systems, our team works closely with customers to fully understand each part's key requirements, including its function, application, relation to other parts, dimensions, and ability to withstand certain elements, pressures, and chemicals.

With more than 60+ years of experience, our engineers have the skills and creativity needed to create any type of custom mold.

With the answers to these questions in hand, the engineering team is then able to determine the key design elements needed to ensure that the injection process will work correctly. These elements include wall thickness, rib design, boss design, corner transition, weld line, and gate/vent placement — all of which are taken into account to create the ideal custom mold.

Once the initial draft design is complete, we conduct a design for manufacturability (DFM) analysis to ensure that the design meets the highest quality standards. And once the design is finalized, the team can start building and testing the new mold.

Tool Designing and Testing

Without a perfect mold, it's nearly impossible to produce a perfect part. So, at The Rodon Group, we take great care with our toolmaking procedures. Because toolmaking requires a lot of time and must allow for optimal accuracy, this step often represents the most substantial investment in a company's manufacturing process. But precise tooling is critical to the success of any plastic injection molding project.

At Rodon, we specialize in creating high-quality, precision-engineered molds that are built to last. Our toolmaking process focuses on a few main areas to ensure we're creating the perfect mold:

- Solid design and engineering
- High-quality stainless steel mold bases and cavities for optimal performance and durability
- Use of state-of-the-art equipment for precise machining
- Attention to strict quality standards and tight tolerances

We understand that creating a quality precision mold is a significant investment for your company, so we make a concerted effort to maximize your return on investment (ROI) by optimizing the following features.

Core Metal

At The Rodon Group, all of our molds on hand use stainless steel as the core metal. For decades, these molds have been able to withstand the pressure of long production runs while still meeting close tolerances.

Number of Cavities

We look to maximize the number of cavities in the mold to maintain the highest level of productivity.

The Mold Base

We estimate the cost of the base — which holds all of the mold cavities, inserts, and components together — based on its size, as well as the type of steel used to make it and allow for the necessary level of customization.

Core/Cavity Machining

We customize molds through the careful placement of cores, cavities, ejectors, cooling lines, and so on.

Part Complexity

We determine the cost of the mold — based on part complexity — including the surface finish of the final part, and the number of undercuts required.

In this way, we can ensure that your mold will last for many years to come. After taking all of these key features into account, actual construction of the mold can begin. During the building process itself, our team uses 420-grade stainless steel as our core tooling metal. To maximize toughness, our engineering team crafts every mold with the maximum number of cavities for the highest efficiency possible during production.

To create a mold, we can reverse engineer a design from a supplied part, or create a mold based on the customer's drawing or CAD file. For the toolmaking process, the team uses several different pieces of equipment — including EDM equipment, CNC mills, lathes, and grinding machinery — to create the part. Once the mold has been built, we then can select the appropriate resin.

Resin Selection

Choosing the right material for a plastic part is also critical for ensuring the success of the plastic injection molding process. Over the last couple of decades, advances in polymer science have allowed for a more comprehensive assortment of available resins, leading to an uptick in the use of plastic in various applications, as well as an increase in available plastics with higher strength and endurance.

At The Rodon Group, our manufacturing team has experience working with a wide range of different resins for various types of applications. We work closely with each customer to establish key requirements for the resin. These requirements include the part's desired appearance, such as texture, color, or transparency; the part's strength, flexibility, or rigidity; the part's chemical or environmental resistance; the part's regulatory requirements; and the part's life expectancy. Throughout our company's history, we have developed close relationships with some of the highest-regarded resin suppliers in the United States, so our customers can be sure we're using only the highest-quality, most reliable resins available.

Production and Quality

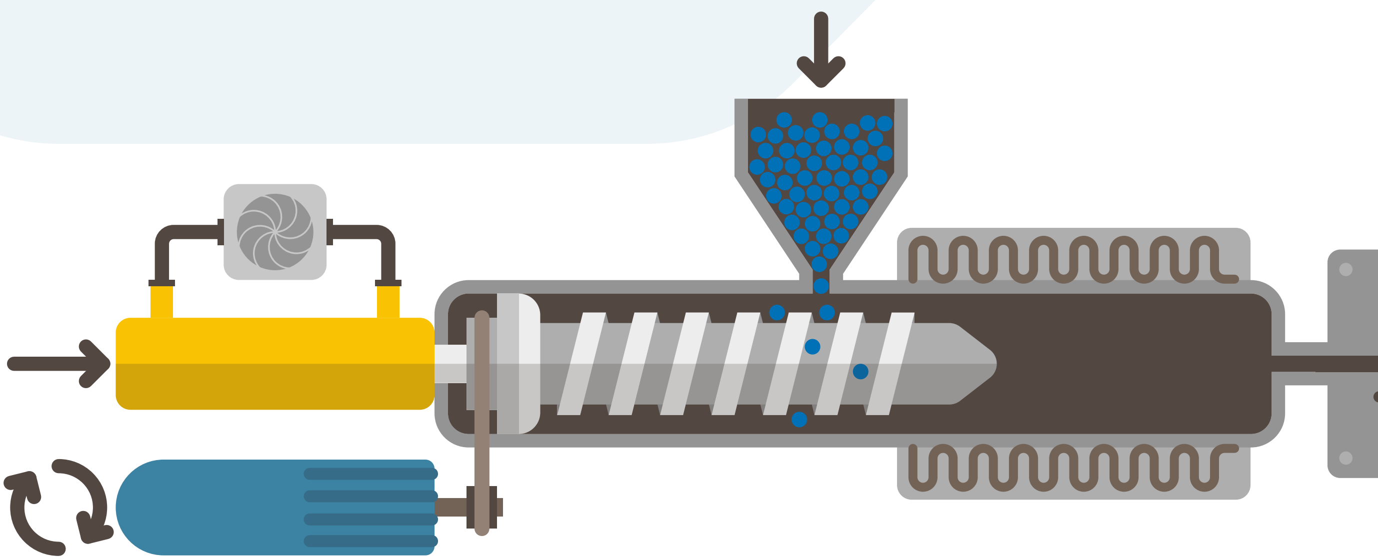
Once the mold has been finalized and the resin chosen, production can finally begin. We house 118 cutting-edge presses in our facility, with varied capabilities and performance options. The team selects which machine to use based on the size of the mold, the number of cavities, and the selected resin.

We then conduct an initial sample run to make sure no defects are occurring. If defects are found, we make corrections to them; if no defects are found, we move forward with production. The team continues to conduct quality checks throughout the entire course of the production process to ensure all products meet the highest of plastic quality standards. Our metrology lab is equipped with state-of-the-art inspection equipment, and we implement a two-step quality inspection process to ensure each of our products meets strict quality standards. We are ISO 9001:2015 certified and are very proud to hold a 99.8% part satisfaction rating among our customers.

At Rodon, we specialize in just-in-time (JIT) manufacturing to monitor and adjust quantities as needed. We also maintain our clients' inventory in-house, meaning they have access to the parts they need when they need them. Rodon utilizes a material requirement planning (MRP) system to track our customers' inventory levels and anticipate any increases in future demand.

Our team values plastic quality above all else, and we're committed to providing the best parts possible - all at the most competitive prices.

Understanding The
Injection Molding Press



With the use of injection molding presses and high-performance resins, plastic injection molders are able to craft a broad range of quality custom parts. Just as all resins are unique, these sophisticated machines offer varied capabilities and performance options depending on press size and type.

To ensure optimal quality and value, it's critical to partner with a manufacturer who can provide the right size press for your specific needs.

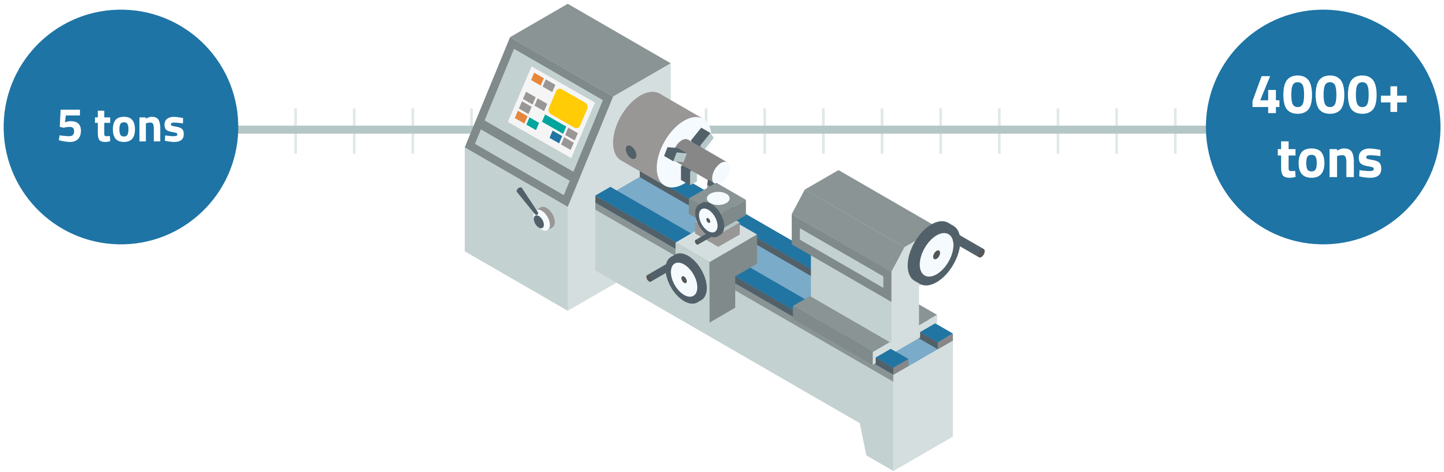
How Are Plastic Injection Molding Presses Rated?

An experienced plastic injection molder can help you select the correctly sized machine for your project, and will be able to provide you with an accurate size estimate right from the start based on a few key pieces of information.

Presses are rated, or classified, based on tonnage, which indicates how much clamping pressure a particular machine can offer. Press tonnage, or force, can range from less than 5 tons to over 4,000 tons. The higher a machines tonnage is rated, the larger it is.

Many plastic injection molders provide a list of molding equipment used in their facilities on their websites, along with each machines' tonnage capabilities.

Press tonnage, or force, can range from less than 5 tons to over 4,000 tons. The higher a machines tonnage is rated, the larger it is.



How Much Clamping Force Do I Need?

Pressure keeps a mold closed during the injection process; too much or too little can compromise quality and result in flashing — the appearance of excess material on the part edge.

To determine the appropriate size press for your application, consider the following key variables:

Press rating

A machine with a 68-ton rating, for instance, will be able to deliver 68 tons of clamping pressure, or force.

Material choice and MFI

The melt flow index (MFI) of plastic, or melt flow rate (MFR), indicates the ease of flow of a molten plastic material. A high MFI will require more pressure than a low MFI.

Size

The size of the part will naturally affect the size of the machine needed. Many calculations include platen size in addition to mold and part size.

Safety factor

A numerical percentage incorporated into size calculations to help avoid defects in the final part, the safety factor acts as a buffer; some experts recommend adding 10% to the overall press size estimate.

For most projects, we recommend calculating required machine size using 2.5 times the surface square inches of the part and incorporating the 10% safety factor. So for example, a part with 42 square inches 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 this plastic injection molded product.

Hydraulic, Electric, and Hybrid Plastic Injection Molding: Which Process is Right for You?

Design engineers have various options when choosing a plastic injection molding process to best suit their specific application. Each of the three primary methods — hydraulic, electric, and hybrid — feature unique benefits and drawbacks. To make the right selection for your project, it's important to have a full understanding of how these methods differ and what they can offer you.

Hydraulic Plastic Injection Molding

First coming into existence in the late 1930's, hydraulic plastic injection molding machines (IMM's) once dominated the market, but their dominance has been impacted since the introduction of all-electric machinery in the 80's. However, hydraulic machines may still be the best option for the job depending on your specific needs, electricity costs, and personal preferences.

These types of machines employ hydraulic cylinders to clamp together two halves of a mold at high pressure. Plastic substrate pellets are then melted, and the liquid is injected into the mold cavity. Once the plastic has cooled and hardened, the mold halves are separated, the part is extracted, and the process is repeated.

To prevent the mold from being pushed open by injection pressure causing excessive material to "flash" around the parts, the clamping unit must be able to supply enough locking force to keep the mold shut during injection. Roughly 3-4 tons of clamp force per square inch of cavity is needed for parts with thin wall sections and deep draw depths. For thick wall sections and shallow draw depths, about 2 tons per square inch is needed.

Today's hydraulic IMMs are able to control clamp forces up to and exceeding 8,000 tons and can create parts weighing in at more than 50 pounds. Hydraulic molding is a popular choice for the automotive industry, which requires the production of large, heavy parts such as bumpers.

Advantages

Hydraulic injection molding is the preferred option for actuating core pulls, ejectors, and valve gates, as well as thick-walled parts that require long hold times. Some of its benefits over all-electric machines include:

- Greater clamp force for large parts
- Better injection rates
- High resistance to wear and tear
- Larger shot size
- Better ejection capability
- Available with gas accumulators to make up for slower clamp movements
- Lower initial purchase price
- Low cost and high availability of replacement parts, resulting in lower maintenance costs
- Easier to control for high-level projects
- There are many units available on the used market because of this method's popularity

Disadvantages

Because they're so powerful, hydraulic machines consume large amounts of energy even when idle. While a typical electric machine may consume about 2.55 kWh during an injection molding process, hydraulic machines may consume 5.12 kWh. They require higher temperatures for molding and more time for cooling, and they're not ideal for clean rooms because of the risk of fluid leakage.

Hydraulic IMMs are also noisier and less precise than all-electric presses. However, as clamp forces increase and control systems improve, hydraulic machines' precision performance also improves.

Electric Plastic Injection Molding

Introduced in 1984 in Japan, all-electric injection molding machines are relatively new to the market but, after rapid adoption, now account for over half of all IMM machines sold in the United States.

These machines are powered by digitally controlled high-speed servo motors rather than hydraulics, allowing for a faster, repeatable, more precise, and energy-efficient operation. Electric machine operation is highly predictable, so once a desirable injection process has been reached, it can be replicated very consistently, resulting in higher quality parts. Once a job has been programmed into an electric machine, its digital controls allow it to run virtually unattended, which lowers labor costs and boosts profits.

Advantages

Since it poses no risk of oil contamination, electric injection molding is well-suited for cleanroom applications. Because of its high precision, this process is also ideal for small- to medium-sized parts and medical products such as Petri dishes and syringes. Other advantages include:

- Tight precision and repeatability with reduced scrap rates
- Cleaner process that never leaks fluid
- Lower downtime commonly associated with hydraulics
- Energy savings from 30% to 70%
- Quieter operation; reduced motor noise below 70 dB
- Higher rapid injection speeds up to 800 mm/sec and faster clamp motion
- Shorter startup time and up to 20% faster cycle times
- Lower unit cost with less material waste
- Requires no consumables such as oil that would need to be replaced or cleaned
- Lower power requirements result in significantly lower operating cost

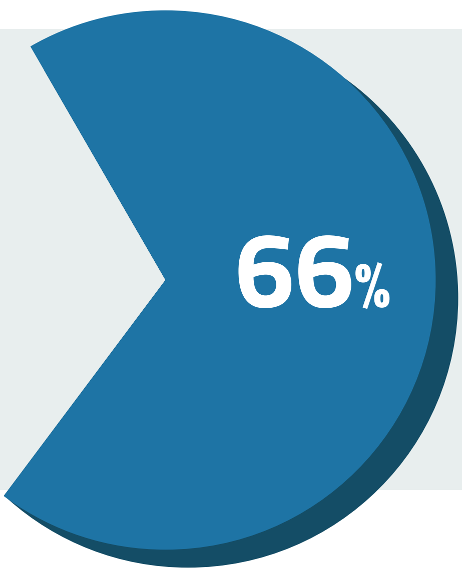
Disadvantages

Although electric machines are faster, cleaner, and more energy efficient, they are unable to achieve the clamp forces produced by hydraulic machines and also come with a higher initial cost. They are toggle-clamp machines that are driven by ball-screws, both of which are wear items, and can be expensive to replace or difficult to find on the used market. And, despite lending great precision, the high positioning accuracy of electric machines can make them less forgiving when compared to hydraulic presses.

Hybrid Plastic Injection Molding

Combining the best of both worlds, hybrid injection molding machines have been on the market for a few decades now, and combine the superior clamping force of hydraulic machines with the precision, repeatability, energy savings, and reduced noise of electric machines. This allows for better performance for both thin- and thick-walled parts. These machines have become increasingly popular over the last few years due to their efficiency and ease of use. Here at The Rodon Group, we recently added seven hybrid presses to our lineup, and now have 23 total.

Nearly 66% of energy in a hydraulic press goes toward screw recovery. So while hybrid presses are more costly upfront, their electrified screw rotation can result in significant savings, which can then be passed on to customers. Our hybrid machines have all the force delivered by full-hydraulic machines, with virtually the same energy-efficiency of an all-electric.



Nearly 66% of energy in a hydraulic press goes toward screw recovery. So while hybrid presses are more costly upfront, their electrified screw rotation can result in significant savings.

Our hybrid machines have all the force delivered by full-hydraulic machines, with virtually the same energy-efficiency of an all-electric.

Advantages

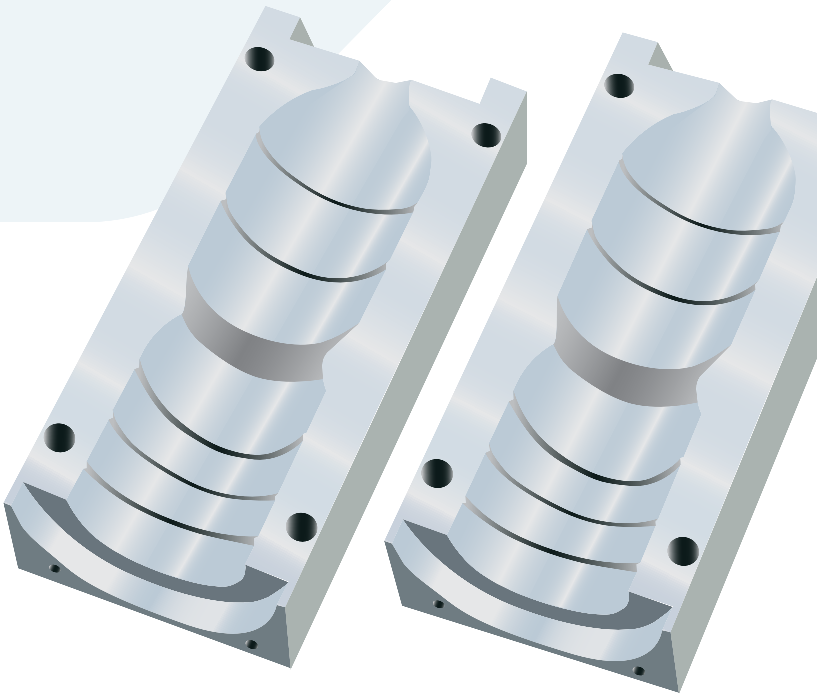
Hybrid plastic injection molding is an energy efficient and long-lasting process without as much demand, all of which leads to a faster return on investment. Some of their specific advantages include:

- Continuous adjustments allowed by the servo pump
- Diversity of product design
- Tends to use a two clamp system over toggle
- Median upfront cost of all three options (hydraulic, electric, hybrid), but could provide the most long-term savings
- Closed loop process with faster response time
- Lower temperature requires less cooling and enables longer oil and machine life
- Faster time to ROI because of these efficiencies

Disadvantages

Because all hybrids are different, it can be difficult to match the correct press to the product, as well as secure replacement machines for certain applications. And if any maintenance is necessary for a hybrid machine, the supervisor must have knowledge of both hydraulic and electric presses.

Cold Runner vs. Hot Runner Systems



Ever since its introduction in the late 19th century, plastic injection molding has revolutionized the way we create plastic products. Although the technology has evolved significantly over the years, many injection molds today still fall into two main categories: hot runner and cold runner systems. Each of these systems has their own benefits and limitations which make them better suited for specific applications.

Understanding the differences between these technologies can help you have a more productive and informed discussion with your plastic injection specialist to determine the most feasible option for your unique application.

Cold Runner Molds

Cold runner molds usually consist of two or three plates that are held within the mold base. The molten thermoplastic is first injected into the mold from a nozzle via the sprue, which fills the network of runners that lead to the mold cavities. In this system, the runners are unheated and act as a delivery system that distributes the molten plastic to the individual molding cavities. The cold runner system subsequently cools the sprue, runner, and gate along with the molded part.

In two-plate molds, sprue and the runner system remain attached to the final product. An ejection system is then used to separate the molded component from the core half of the mold. For those of you who assembled a model car at some point in your youth, the runners and the parts were not separated. The child assembling the model was responsible for that final part of the process.

In three-plate molds, a stripper plate contains the sprue which delivers the molten thermoplastic to the network of runners which are contained in a separate cavity plate. The final molded part is formed and cooled on the core half of the plate without any runners or gates attached. The three-plate mold system automatically separates the runner from the part as the mold opens. In both the two-plate and three-plate mold systems, the runner may be reground and recycled, thereby reducing plastic waste.

Hot Runner Molds

Hot runner molds consist of two plates that are heated with a manifold system. The manifold helps maintain a consistent temperature by keeping the molten thermoplastic in the runners at the same temperature as the heating cylinder. The heated runners deliver the molten plastic to nozzles that fill the core mold to form the final part.

The heated runner system is housed in a separate plate. This plate remains stationary during the molding cycle. The core molding plate then opens to reveal the final molded part without any runners attached.

There are several types of hot runner systems. In general, however, they fall into two main categories; externally heated and internally heated. The externally heated systems are well suited to polymers that are sensitive to thermal variations. Internally heated systems offer better flow control.

The hot runner process eliminates runners entirely, so recycling and regrind (which can only be done with virgin plastics) do not impact cycle times. A variation on this system is known as an insulated runner. The insulation, rather than heat, keeps the plastic in a molten state. This system can only accommodate a few types of plastics—specifically semi-crystalline polymers, which tend to have a low thermal conductivity.

Advantages & Disadvantages of Each System

Cold runner systems are, typically, relatively more cost effective than their hot runner counterparts. The total cost to produce a molded item and lower general maintenance expenditures both add to the overall reduction in cost. They are also capable of handling a wide variety of commodity and engineered thermoplastics. Additionally, cold runners are more flexible than hot runners since runner and gate locations can be easily changed or upgraded, allowing for faster design changes.

However, one of the significant disadvantages of cold runner systems is the actual production of the runner itself. In two-plate cold runner systems, the runner needs to be manually separated from the finished part after each run. If these runners are not reground and recycled, the plastic material is wasted after each run. Furthermore, the removal of the sprues and runners – and the time it takes to recycle each runner – adds to the cycle time, resulting in a slower production time than hot runner systems.

- ✓ Cold runner systems are, typically, more cost effective than hot runner systems.
- ✓ Cold runner systems are capable of handling a wide variety of commodity and engineered thermoplastics.
- ✓ Cold runner systems are more flexible than hot runners since runner and gate locations can be easily changed or upgraded.
- ✗ In cold runner systems the runner needs to be manually separated from the finished part after each run.
- ✗ Slower production time than hot runner systems.

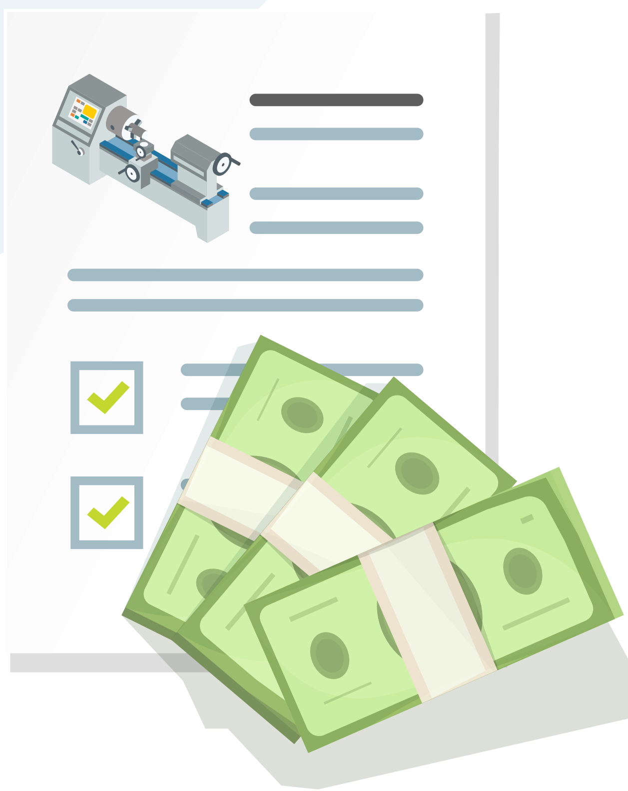
Hot runner systems generally possess significantly faster cycle times than cold runner systems. The elimination of the runners greatly reduces the number of post-production activities such as runner and sprue removal, regrinding, and recycling. The lack of runners also reduces potential waste during the molding process and eliminates the need for robotics to remove runners. This contributes to an overall cost reduction and increases the efficiency of the mold automation process.

Also, because each runner feeds a dedicated mold, hot runner systems are capable of fabricating larger parts. Hot runners, however, tend to be more expensive than cold runner injection mold systems. Because each hot runner feeds into a dedicated mold, additional internal or external heating sources are required to keep the runners at a consistent temperature.

Additionally, the tooling cost is usually higher than that of cold runner systems. As a consequence, molded parts can be more expensive to produce, thereby increasing the cost of the total production volume. The complex nature of the hot runner equipment also means that it requires more careful inspection and maintenance, leading to an increase in maintenance costs and potential downtime. Also, internally heated hot runner systems may not be suitable for thermally sensitive polymers since there are areas where material could stick to the surface and become overheated.

- Hot runner systems generally possess significantly faster cycle times than cold runner systems.
- Hot runner systems allow for overall cost reduction and increased efficiency of the mold automation process
- Hot runner systems are capable of fabricating larger parts.
- Hot runners tend to be more expensive than cold runner injection mold systems.
- The tooling cost is usually higher than that of cold runner systems.
- Hot runners have increased maintenance costs and potential downtime compared to cold runner systems.
- Internally heated hot runner systems may not be suitable for thermally sensitive polymers.

Determining The Cost of an Injection Mold



One of the most common questions for those looking to source a plastic injection molder is “How much will a plastic injection mold cost?” It’s one of the most important questions, since the actual mold represents the most significant expense in upfront production costs. That being said, many factors go into determining the full cost. With any custom injection molding project, your injection molder should be able to give you the final price tag. In this article, we will review the variables that can impact the cost so that you can be better informed when making a mold purchasing decision.

The Merits of Building a Mold in the U.S

The Rodon Group has been in this business long enough to have worked with, or attempted to work with, molds that were created in Asia (most often, China). Many of these molds are not built to meet strict tolerances, since they don't have to be. As labor costs rise in China, this no longer becomes an affordable manufacturing process. In addition, molds made in other countries are made from lesser quality steel or aluminum. Again, this causes quality issues and shortens the life of the mold.

Most high-production molders in this country use only the best stainless steel available for the mold base and cavities. They utilize the latest technology in CNC machining and have seasoned tool makers who know how to create molds that meet the highest performance standards. When sourcing companies, be sure to look for U.S. based manufacturers.

When sourcing companies, be sure to look for U.S. based manufacturers.

Comparing Quotes

Procurement and purchasing managers have the unenviable task of obtaining quotes from a few injection molders for each project. Depending on the input (in terms of drawings, prototypes or sample parts), the cost quotes can vary greatly. Designers should also look at all of these inputs and determine the best molding solution. They may redesign the part to maximize manufacturing efficiency and increase the number of components that can be made with each molding cycle.

Generally, molds made with tighter tolerances, more cavities, and a longer production life will take longer to build and will cost more upfront.

The savings with a high-quality mold are long-term. These molds require less maintenance and last longer than lower quality molds.

Variables That Impact the Cost

The core metal:

For shorter production runs, some mold makers will use molds made from aluminum. This is a perfectly reasonable choice if you will not need the mold to perform long-term. However, if a project requires that a mold last for several years, an aluminum mold may cost more in the long-run.

The number of cavities:

It is pretty intuitive when you think about it. Fewer cavities in a mold require less tooling work and 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.

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. Most mold bases come in standard sizes and are further machined to meet the requirements of a specific project.

Core/Cavity machining:

All molds must also be customized. Customization includes the placement of cores, cavities, ejectors, and cooling lines. 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 Injection Molders

Some mold builders also manufacture the parts. This type of integration can help defray the mold building cost. Often full service, turnkey 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.

The cost of a quality injection mold is undoubtedly a significant expense. However, tight-tolerance, precision molds that are made from the best steel available should last for years to come. The upfront cost must be calculated or amortized into the lifetime value of the project. Will these parts be in production for several years or several months? Does the project require a high-volume of parts? Are faster cycle times needed? If you answered yes to these questions, then the initial investment in a quality mold will lower the per part cost and will end up saving money in the long-run.

We hope this overview of key cost factors helps you in getting and comparing quotes for your future projects. Working with a reputable on-shore molder, with a long history of mold building for various industries, is a great place to start your bidding process.

The cost of a quality injection mold is a significant expense. However, tight-tolerance, precision molds that are made from the best steel should last for years to come.

Common Injection Molding Defects & How to Avoid Them

When working with any manufacturing process, a number of defects unique to that process commonly occur. This is true across many processes and industries, including injection molding and high volume injection molding.

As we have previously discussed on this blog, there are several common injecting molding defects; however, an injection molder who is vigilant about quality, like our team at The Rodon Group, will be able to manage these injection molding defects, minimizing or eliminating them all together.

These six most common plastic part defects can all be traced to one of three sources: the resin or additives used, the injection molding process, or the mold itself.

Resin and Additive Caused Defects

Two common defects caused by issues with the resin or resin additives used during injection molding are delamination and discoloration.

Delamination

Delamination, when a finished part has a layer of flaky material at the surface, hurts both aesthetic of your part and its strength. Caused by moisture contamination of the resin pellets or by other contamination of the melted resin with a dissimilar resin, or by release agents in the mold, delamination is the result of the resin being prevented from bonding.

A number of methods, both simple and more complex, can be used to prevent delamination. If moisture is the issue, pre-drying the resin pellets or increasing mold temperature will help. If mold release agents are the cause, a mold redesign that places more focus on the ejection mechanism will help to eliminate mold release. If it is caused by cross-contaminated resins, that will need to be replaced with virgin material.

Discoloration

Discoloration is simply when a finished part is a color different than intended. Caused most commonly by leftover pellets in the hopper, too hot of a barrel temperature or leftover resin in the feed zone, the problem can be addressed by thoroughly flushing the hopper and feed zone of a machine in between processes, thus preventing discoloration as a matter of course. Purging compound can also be effective to remove unwanted color or resin.

Process-Caused Defects

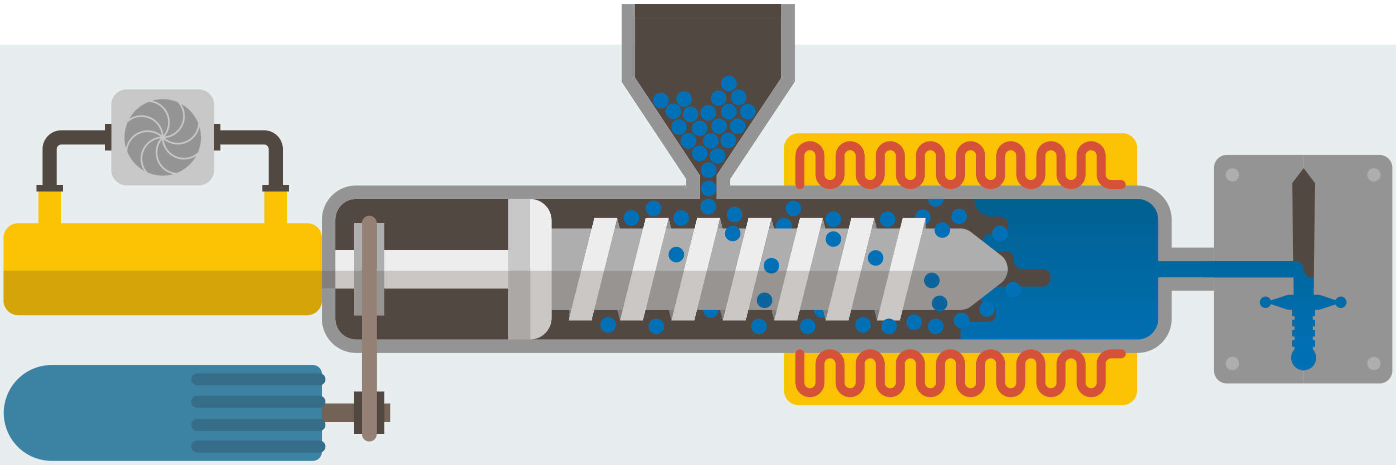
Despite continual advances in injection molding technology, process-derived injection molding defects still occur. Two of the most common are burn marks and flow marks.

Burn Marks

Burn marks are surface marks, sometimes advancing to degraded plastic, that are caused by either trapped air which becomes overheated or actual resin that overheats. There are three ways to avoid burn marks: decrease resin injection speeds, which will lower the probability of air becoming trapped; include or optimize venting and degassing systems; or reduce the mold and/or melt temperature.

Flow Marks

Flow marks are lined patterns, often wavy, or discoloration on a part surface. They are most commonly caused by resin cooling too quickly or improper gate location. In the best case scenario, flow marks can be eliminated by increasing injection speed and pressure, which will help to ensure uniform filling and cooling. In a worst case scenario, a mold redesign with an emphasis on avoiding sudden flow direction changes and gate location may be necessary.



Mold-Caused Defects

Flash and short shots are two of the more common injection molding defects caused by mold design or maintenance issues.

Flash

Sometimes known as burrs, flash is the occurrence of thin, wafer-like protrusions on a finished part caused when melted resin escapes the mold cavity. Most common along the parting line or up an ejector pin, flash can be caused by excessive injection speed or pressure, in which case the fix is a simple reduction.

More often flash is due to poorly designed or severely degraded molds, in which case a redesign or retooling is required. Flash can also be caused by too high of a mold temperature and excessive barrel heat.

Short Shot

A short shot is literally when a shot of resin falls short of filling the mold. It can be caused by attempting to use the wrong resin type or by poor process settings, but is most commonly caused by gate blockages or too small of a gate diameter, a common problem due to too low pressure or not enough heat. If a higher melt index resin or increased melt temperature doesn't solve a short shot problem, you may need to redesign the runner system to optimize flow.

Glossary Terms to Know

Plastic injection molding has a language all its own, and with hundreds of unique terms, it can be difficult to learn the language. To help, we put together a list of the 15 more commonly used terms to know when discussing plastic injection molding, mold parts, machinery, materials, and problems. We hope you find this to be a useful resource.

Injection Molding Terms:

Resin

Resin is the raw material used to create the finished part in the plastic injection molding process. With hundreds of commodity and engineering resins available on today's market, the material selection process for plastic injection molding may seem daunting at first, so research your options carefully, and consult with an experienced plastic injection molder to help determine the ideal choice.

Colorant

A pigment system, usually in pelletized form, or liquid, which is mixed with resin to produce the desired color. To pinpoint the desired color for each plastic product or part, a color matching process must be completed, which allows engineers to develop a specific color concentrate for a particular application. Typically, a chip, plaque, or Pantone number provides an approximate idea of the desired hue, and information about the specific polymer being used helps to determine the formulation for the color concentrate.

Mold

A hollow form often made from stainless steel that plastic is injected or inserted into to manufacture a plastic part. Molds can be expensive to design and manufacture because of this they are typically only used in mass production. As one of the most significant production investments, it is critical that the molds are made with a great deal of accuracy. Tight-tolerance, precision molds that are made from the best steel available should last for years to come.

Mold Cavity

The hole in the mold that is in the shape of the desired part, this is where the plastic resin is injected into to make the part. Fewer cavities in a mold require less tooling work, time and ultimately less cost. A reputable, experienced molder will be able to maximize cavitation 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 different parts. They tend to produce inferior products and have more downtime due to maintenance issues.

Wall Thickness

Of all the various design aspects, wall thickness has the most significant impact on the cost, production speed, and final quality of a part. Wall thicknesses are not subject to any restrictions, but generally, the goal is to create the thinnest wall possible while taking into account the part's structural requirements and overall size and geometry. The flow behavior and material qualities of the resin should also be considered. Uniform wall thickness also allows for the most efficient, uniform flow of resin through a tool for ideal processing. Variations in wall thickness cause molten polymers to take preferential flows, leading to air trapping, unbalanced filling, and weld lines.

Flash and Burrs

Sometimes known as burrs, flash is the occurrence of thin, wafer-like protrusions on a finished part caused when melted resin escapes the mold cavity. Most common along the parting line or an ejector pin, flash can be caused by excessive injection speed or pressure, in which case the fix is a simple reduction. More often flash is due to poorly designed or severely degraded molds, in which case a redesign or retooling is required. Flash can also be caused by too high of a mold temperature and excessive barrel heat.

Runner System

The channel system that allows the flow of the melted material to fill the part cavities. There are two main categories: hot runner and cold runner systems. Each of these systems has its benefits and limitations which make them better suited for specific applications. Understanding the differences between these technologies can help you have a more productive and informed discussion with your plastic injection specialist to determine the most feasible option for your unique application.

Hydraulic Process

Hydraulic is the predominant type of process used in plastic injection molding. These types of machines employ hydraulic cylinders to clamp together two halves of a mold at high pressure. Plastic substrate pellets are then melted, and the liquid is injected into the mold cavity. Once the plastic has cooled and hardened, the mold halves are separated, the part is extracted, and the process is repeated. The hydraulic process uses a hydraulic press which is not as precise as electric presses.

Electric Process

All-electric presses were introduced in 1983. The newer all-electric press technology is quieter to operate, faster and have higher accuracy. These machines are powered by digitally controlled high-speed servo motors rather than hydraulics, allowing for a faster, repeatable, more precise, and energy-efficient operation. Electric machine operation is highly predictable, so once a desirable injection process has been reached, it can be replicated very consistently, resulting in higher quality parts. The machinery required for the all-electric process is more expensive than the hydraulic process.

Hybrid Process

Combining the best of both worlds, hybrid injection molding machines have been on the market for a few decades now, and combine the superior clamping force of hydraulic machines with the precision, repeatability, energy savings, and reduced noise of electric machines. This allows for better performance for both thin- and thick-walled parts. These machines have become increasingly popular over the last few years due to their efficiency and ease of use.

End-of-Arm Tooling

Speed and efficiency in plastic injection molding equate to cost savings. So, it is no surprise that robots play a significant role in improving the manufacturing process. From simple sprue pickers to complex automated End-of-Arm Tooling (EOAT), the industry is taking advantage of this technology. The EOAT is often assembled along with the electronics, pneumatics, and sensors needed to meet the specific processing requirements of the job.

Tonnage

Presses are rated, or classified, based on tonnage, which indicates how much clamping pressure a particular machine can offer. Press tonnage, or force, can range from less than 5 tons to over 4,000 tons. The higher a machines tonnage is rated, the larger it is. Pressure keeps a mold closed during the injection process; too little can compromise quality and result in flashing — the appearance of excess material on the part edge. To determine the appropriate size press for your application, consider the following key variables such as material choice, size of the part and press rating.

Hopper and Barrel

The hopper stores the plastic to be used in the injection molding process. For materials such as Nylon, ABS, and PET, a dryer unit may be added to dry the plastic for processing and to keep any external moisture away from the material. The hopper may also contain small magnets to prevent any harmful metallic particles from entering the machine. The plastic that is placed in the hopper is usually in some type of granular form. The plastic material is then melted using heater bands and is then injected through the nozzle into a mold cavity. The pellets are fed from the hopper to the barrel where the material is then melted in a controlled fashion and injected into the mold in the machine.

Working with The Rodon Group

The Rodon Group is an ISO 9001:2015 certified, high volume (we're talking millions!) plastic injection molder. In business since 1956, we make billions of parts each year in our 125,000 square foot facility. We offer a turnkey manufacturing solution including mold design, mold building and high volume parts manufacturing. Our globally competitive prices eliminate the risks of sourcing offshore.

Every member of our team works together to hone in on your project's success, crafting the ideal finished product for your specific needs. By trusting The Rodon Group to do the heavy lifting, your in-house team gets more than just a superior part with exceptional performance and great value — they also maintain the ability to focus on their day-to-day jobs without interruption.

With over 125 injection molding presses and over 65 plus years in the business, our company offers the highest-quality, most reliable resources and equipment available. Material knowledge and specialty sourcing, project strategy, custom design solutions and tooling, state-of-the-art technology, and a long-standing dedication to quality are all guaranteed with Rodon.

About

THE RODON GROUP®

Contact Us Now | View our Resource Library | Visit our Blog

2800 Sterling Drive Hatfield, PA 19440 | 1-800-523-8792 | info@rodongroup.com

www.rodongroup.com