The Ultimate Guide to Injection Molding

THE TECHNOLOGY HOUSE

Everything You Need to Know About Injection Molding

In today's ever-evolving world of manufacturing, plastics are being used to make everything from electronics and medical equipment to automotive components and kitchenware.

To create critical components and ensure optimal performance, many manufacturers make use of injection molding.

When businesses need high volumes of intricate parts, they turn to plastic injection molding. This process provides a high level of production while minimizing the cost per unit.

In this handbook, you'll learn how to use injection molding to efficiently create hundreds or thousands of custom parts in many shapes, sizes, and materials.

Table of Contents

What Is Injection Molding?

Injection molding is a great manufacturing technology that produces large amounts of parts repeatedly at a low cost. Plastic injection molding uses a combination of thermal principles and pressure to create the parts and components you need for your business.

During plastic injection molding, a plastic material is melted into a liquid state and then forced into a mold under high pressure. The part quickly cools and solidifies to create the unit.

ADVANTAGES OF USING INJECTION MOLDING

Plastic injection molding offers a wide range of unique benefits. It allows companies to produce parts in complex shapes that would be too difficult or costly to machine otherwise, and it also allows many plastic parts to be created at the same time, making it ideal for high-volume applications.

Other key advantages include:

- Low labor costs
- High-repeatability
- Minimal required work after molding

The injection molding process also produces very little waste, since the unused plastic scrap can be easily recycled and used in other molding products.

Injection Tooling and Molding

All tools can include automation, slides, multi-cavities, and/or family setups. Here are the different types of tools available:

RAPID 3D PRINTED TOOLING

This tooling option is used when companies need parts very fast for testing, prototyping, or production. Tolerances and the number of parts produced varies, and parts are also restricted by size and material. Rapid 3D printed tooling is typically ideal for small parts comprised of low-temperature materials (TPU, ABS, PP, PE, etc.).

PROTOTYPE ALUMINUM TOOLING

This is a great option to get parts fast. Lead times are generally in as little as three weeks depending on size and complexity. Molds are usually good for 2,500 parts or less and can be created in single, family, or multi-cavities. Tools can then be used as bridge or production tools with special coatings.

PRODUCTION ALUMINUM TOOLING

In the niche markets of today, all parts do not need big, expensive steel tools. Over the last 20 years, The Technology House (TTH) has worked with production aluminum tools and Master Unit Die (MUD) inserts that allow for shorter lead times and lower costs because molds can last for 10,000 to 20,000 shots.

PRODUCTION STEEL TOOLING

When tolerances are tight and volumes are high, the best option is steel tooling. Steel tooling gives you the confidence of consistency from part to part at high volumes. You could also design the tool to be steel safe. This involves creating the mold slightly out of tolerance so you can sample the tool, measure the parts, and then adjust the tool to “creep up” on the tolerance. This avoids overshooting a dimension. This is common for tight tolerances or high-shrink materials. In general, a typical tight tolerance for injection molding is +/-0.002". This is a great option when you need multiple cavities and fast cycle times to get lots of parts quickly and at low costs. Typically, this is a great option for tooling if you expect volumes to exceed 20,000 parts.

8 Questions to Think About When Choosing Your Injection Mold Tooling

Injection molding is one of the most common ways to manufacture products. The first step in injection molding is to choose a tooling option that works for your project. Here are eight questions that will help make it a little easier to choose your injection mold tooling.

1. WHAT IS THE SCOPE OF THE PROJECT?

This is probably the most important factor in determining which tooling method to use.

If the part is for pre-production, then the answer is simple: aluminum tooling. This is common when the project requires the part to be made with production injection molding material. Aluminum tooling offers lower costs and faster lead times. If the material requirement isn't needed, see alternative RTV Molding and 3D Printing options.

If the part is for production, then there are a couple of things to consider. What is the part's Estimated Annual Usage (EAU)? How long will the project run?

2. HOW DOES PART SIZE AFFECT TOOLING?

Part size plays a big factor in determining tooling. Larger parts will need to be built in a standalone tool. However, smaller parts that fit within the size parameters may be subject to a (more efficient) cheaper alternative.

Insert tools are extremely popular for smaller components. Instead of paying the full price of a standalone tool, TTH can look to build an insert tool that fits into the (standard MUD) base unit on our press.

For example, if you have a small 3"x 2"x 1/2" housing that needs to be produced via injection molding, TTH will build an insert tool out of aluminum or steel that fits into pre-existing bases on our press instead of building a full standalone tool for the small part. This is an extremely economical, waste-minimizing method of producing smaller components. TTH offers insert sizes ranging from a 5"x 5" all the way up to an 11"x 14".

3. HOW DO PART VOLUMES AFFECT TOOLING?

Part volumes can affect tooling, especially when volumes are greater. The standard is using a single cavity tool for low volumes of a couple hundred or a couple thousand parts per year.

When part quantities and the life of the project are unknown, or there is no solid forecast, single cavity tools are a good place to start. You can always consider building multi-cavity tools later on. Multi-cavity tools come with a little more upfront cost, but they can significantly lower part piece prices.

8 Questions to Think About When Choosing Your Injection Mold Tooling (cont.)

4. DOES PART MATERIAL AFFECT TOOLING?

Yes, it does. The part material has a direct effect on tooling for a couple of reasons. Mild injection mold resins like polypropylene are a lot easier on a mold, therefore they contribute to a longer tool life. Harsher injection mold resins like a glass-filled nylon wear down a tool much more quickly. This can be a crucial deciding factor when your part has a life of 8,000–12,000 pieces, and you're deciding between aluminum or steel tooling.

5. DOES PART GEOMETRY AFFECT TOOLING?

Yes, it does. TTH thoroughly evaluates each part before quoting, looking at part features that will affect the tool. Does it have undercuts or cores? What are its surface finish requirements? Will it be polished or textured? These factors all affect the type of tooling used.

6. WHAT IS THE LIFE EXPECTANCY OF A TOOL?

Aluminum tools have lifespans of 2,000–10,000 parts depending on the type of aluminum used, part material, and geometry.

Steel tools have lifespans of more than 100,000 parts depending on the material and geometry of the part. All tools may need maintenance after it's been running production for awhile.

7. HOW LONG DOES IT TAKE TO BUILD A TOOL?

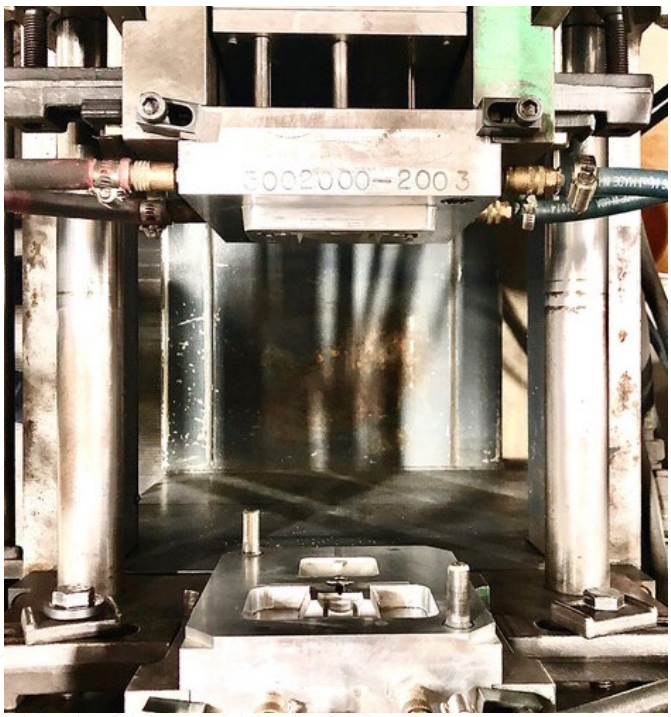
This changes on a part-by-part basis, but generally:

Aluminum tools can be built in anywhere from 4–6 weeks for small parts and 6–10 weeks for larger parts

Steel tools can be built in anywhere from 6–8 weeks for small parts and 8–12 weeks for larger parts

8. WHAT IS THE COST DIFFERENCE FOR INJECTION MOLD TOOLING?

This also changes on a part-by-part basis, but typically a steel tool costs anywhere from 20–30% more than an aluminum tool.



Injection Molding Materials

One of the many positives associated with plastic injection molding is the wide variety of material options available to manufacturers and designers. It's easy to become inundated in the material decision-making process since different industries have different needs when it comes to virgin, recycled, UL-rated, FAR-rated, and food-grade materials.

Some of the more common injection molding materials used include:

- Nylon
- PC/ABS blends
- Polycarbonate
- Polypropylene
- Polyethylene
- PBT
- Glass fiber nylon
- Polyvinyl chloride (PVC)
- High-density polyethylene (HDPE)
- Acrylonitrile butadiene styrene (ABS)
- Acetal (Delrin)
- Soft durometer materials (including Santoprene® and Sylvin®)

Mold Finishes

Types of Finishes	Current SPI Finish Numbers		Description and Previous Finish Numbers		Roughness Comparison R.A.
Diamond	#3 Diamond Buff	A1	is comparable to	#1	0–1
	#6 Diamond Buff	A2	is slightly finer than	#2	1–2
	#15 Diamond Buff	A3	has more imperfections than		2–3
Paper	600 Grit	B1	is finer than	#3	2–3
	400 Grit	B2	is slightly finer than		4–5
	320 Grit	B3	is comparable to		9–10
Stone	600 Stone	C1	is finer than	#4	10–12
	400 Stone	C2	is slightly finer than		25–28
	320 Stone	C3	is comparable to		38–42
Dry Blasted	#11 Glass Bead	D1	is finer than	#5	10–12
	240 Aluminum Oxide Blast	D2	is comparable to		26–32
	#24 Aluminum Oxide Blast	D3	is a little more coarse than		190–230

Designing Molded Parts

There are several essential design elements to consider to ensure successful injection molding:

WALL THICKNESS

One of the most basic design parameters is to keep the wall thickness consistent. Parts with a uniform wall thickness tend to warp less, will fill in properly, and minimize shrink variability. But how much wall thickness is typically allowed? Ideally, there should be no variation, but ribs should only be 60% at its base of the wall it intersects.

GATE PLACEMENT

A part must have a gate, or an opening that allows the plastic to be injected into the mold. The most effective gates are ones that enter the thickest part of the cavity and then flow to the narrower areas. Since the gate will be slightly visible on the part, it's best to have it on a non-cosmetic surface.

RADII

Radii, in particular fillet radii, should be applied at the base of all interior ribs and walls. In general, plastic doesn't like sharp corners for a couple of reasons:

- Radii help with the flow of materials when filing the part.
- Radii help improve the integrity of the part, especially at the base of thin ribs.

Sharp corners are stress risers that can cause part failure. The radii allow the molten plastic to navigate around the corners more easily and reduces the molded stress and the pressure to fill the tool.

DRAFT

Draft is when the side walls in the mold are tapered in the same direction that the mold opens. Draft facilitates the removal of the part from the mold. It's important to note that different degrees of draft are required based on part geometry and surface texture. A tool should use at least one degree of draft for all vertical surfaces (two degrees works very well for most parts). TTH can add various textures to the tool.

Injection Molding FAQs

Q: What is the difference between hard and soft tooling?

A: Soft tooling is typically considered prototype tooling and uses aluminum tools. It's used to get parts ready quickly for production testing. These tools are usually only good for a couple thousand parts. Hard tooling is considered production tooling. Usually, tools are created using steel for better tolerancing, repeatable cycles, reduced cycle times, and extended tool life.

Q: What is MUD or insert set tooling?

A: MUD, or insert set tooling, involves modular tools where the core and cavity are created in different sizes that fit into standard MUD bases. They can be aluminum or steel. This is to reduce mold build times, material costs, setup time, and inventory space. This type of tooling is used for smaller- to medium-sized parts but is good for all different quantities.

Q: What is the difference between a hand load and a slide?

A: Hand loads and slides are pieces of tools that are needed to create the features that are not in the line of the draw when trying to open a tool. In both cases, the tool sections must be removed from the tool before the tool can open.

With a hand load, the operator must physically remove the tool section to open and close a tool for molding. Hand loads should be easily handled and should not be too large or heavy. Using hand loads is a great way to reduce the upfront cost of a tool. However, hand loads increase part cost and tool maintenance, and result in longer cycle times. They're good options for prototypes, bridges, and low-volume tools for smaller- or medium-sized parts.

With a slide, the molding process is done automatically without labor. Slides in tools increase initial tool costs but reduce costs in the long run by cutting cycle times and the risks associated with operator handling. Use slides in your tools when you want to reduce risk and piece prices or you want consistent high volumes.

Injection Molding FAQs (cont.)

Q: How many parts can you make from an injection mold tool?

A: This varies based on multiple factors, including mold type, mold material, part material, and part geometry. As a standard:

- 3D printed tools = 10–100 shots
- Rapid prototype aluminum tools = 2,500 shots
- Production aluminum tools = 10,000–20,000 shots
- Production steel tools = 50,000+ shots

Q: What is overmolding?

A: Overmolding is the process in which a plastic part is molded "over" a substrate part. The substrate part is typically a rigid part in either plastic or metal. We then "overmold" an elastomeric or rigid material onto or around the substrate.

Examples of this are rigid handles with soft grips. To do this, we would create two separate tools, one tool for the substrate and the other tool for the overmold. These can be in the same tool or different tools but must be separate cavities.

Q: When should I use injection molding?

A: Injection molding is best for high-volume production and low-cost parts. Consistent, repeatable parts and processing is required, and chances of design changes are low.

Bring Your Ideas to Life

TTH is a recognized leader in 3D printing and additive manufacturing, as well as CNC machining (especially 5-axis CNC machining services), urethane casting, and injection molding services.

Our custom injection molding department is an agile team that can shift quickly based on your needs to run prototypes, bridge volumes, or high-volume production. We handle projects in all different materials (e.g., Nylon, ABS, PC, and TPU) for many industries (including aerospace, medical, consumer, and industrial) ranging in volumes up to 1,000,000 parts or more.

Our injection mold presses range from 30 tons to 596 tons with barrels sizes of 3oz to 70oz for small, medium, and large parts. With an agile mindset, our team can change tools and materials quickly without the downtime of other facilities.

Our project managers and engineering team work with a "one-stop-shop" mindset to ensure your product is designed for the right process, and if injection molding is best for your requirements, we will make the tooling, material, mold press, and process work for your specific application.

APPLICATIONS

- Low- to high-volume production
- Small to large parts (up to 60oz shot size)
- Wide range of available materials
- Tight tolerance molding
- Validated injection mold process
- Low-cost manufacturing option
- Customized tooling and part production
- Flexible manufacturing to ramp-up production
- No minimum order sizes
- Part design reviews

With every injection mold tooling quote, we provide a design review with recommended changes to reduce cost and increase part quality.

Bring Your Ideas to Life (cont.)

TOOLING AND MOLDING

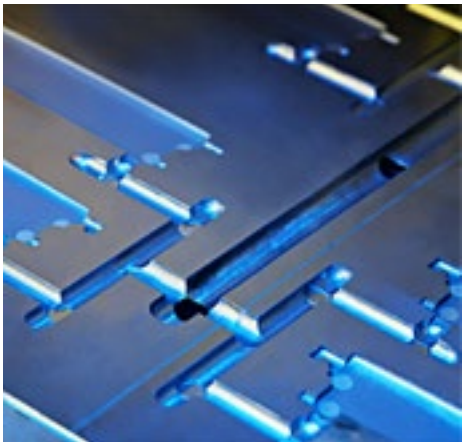
At TTH, we work with our customers to provide tooling options based on the needs of their molded parts. We have domestic and international tooling resources depending on the requirements and needs of our customers, can provide full tools or MUD insert sets, and offer both aluminum and steel material options.

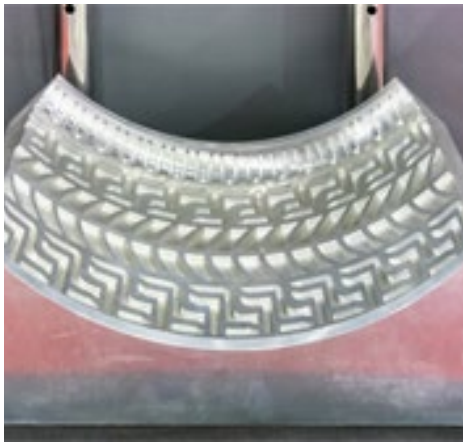
To help design the right tool for the right molding material and injection press, we can conduct a mold fill and flow analysis to better predict what will happen based on our tool design. Our company also has invested heavily in process validation and risk analysis, so we can understand the processing ranges of the parts we mold day in and day out. See the many different value-added processes we use to support production.

CUSTOM FINISHES

For finishing, we can customize your parts by treating the injection mold tooling for optically clear, high gloss, matte, or any of the standard mold textures available. We can also custom color parts in many different ways, including coloring in-house, compounding, or pre-coloring materials. Our team can also add any part decorations required, such as paint, labels, silk screening, or pad printing to show off your brand.

Efficiently create hundreds or thousands of custom parts in many shapes, sizes, and materials with aluminum and steel tooling tailored for your prototype, bridge, or production injection molding project. We can help!





The Technology House

10036 Aurora-Hudson Road
Streetsboro, OH 44241

440-248-3025

tth.com

Contact Us

THE TECHNOLOGY HOUSE