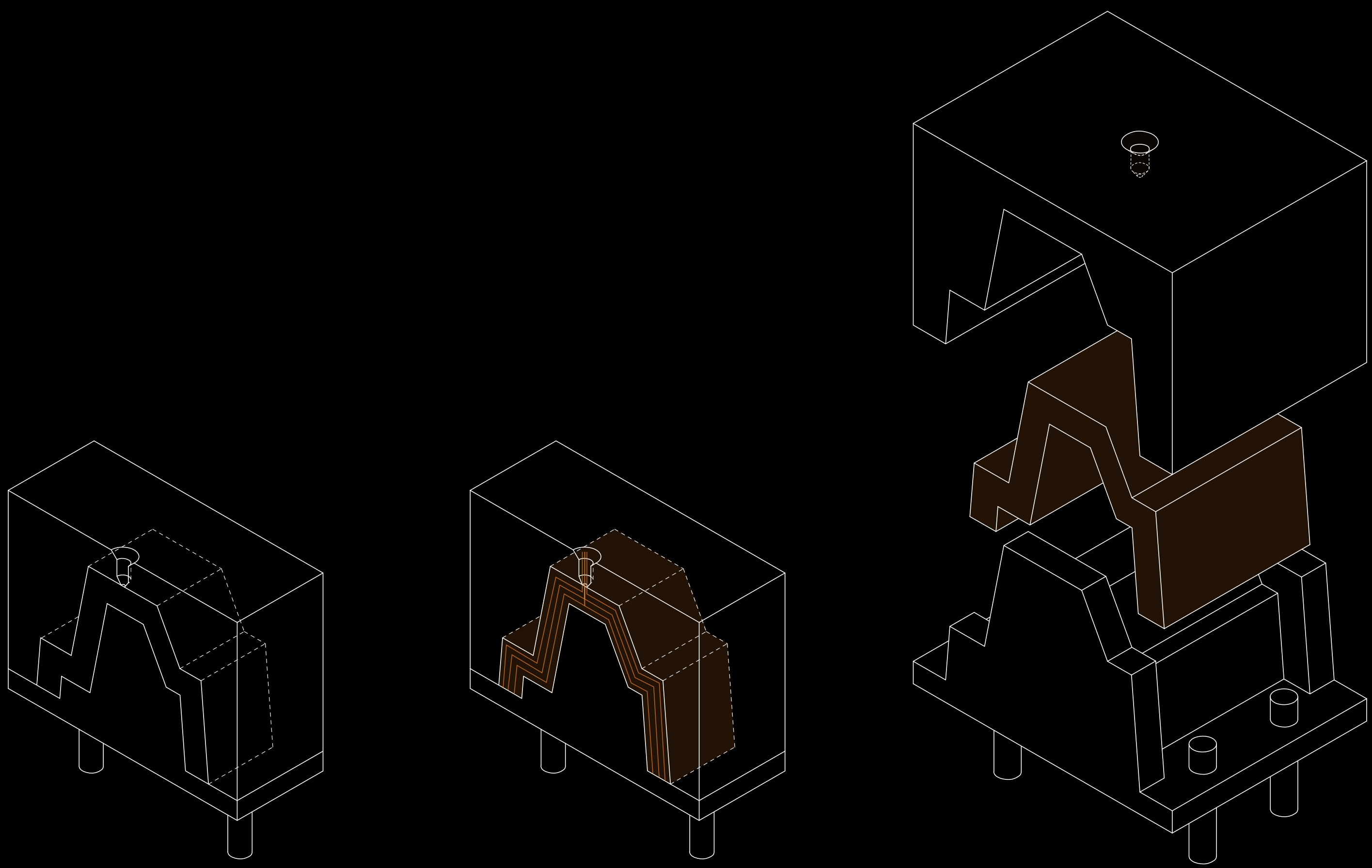


# Injection molding: The definitive engineering guide

This guide has all you need to know about Injection molding and how to get started with the mass-production of plastic parts. Master the basic principles of the technology and learn quickly actionable design tips that save time & cut costs.

# Table of Contents

Part 4

## Cost reduction tips



Part 5

## Start Injection molding



Part 6

## Useful Resources

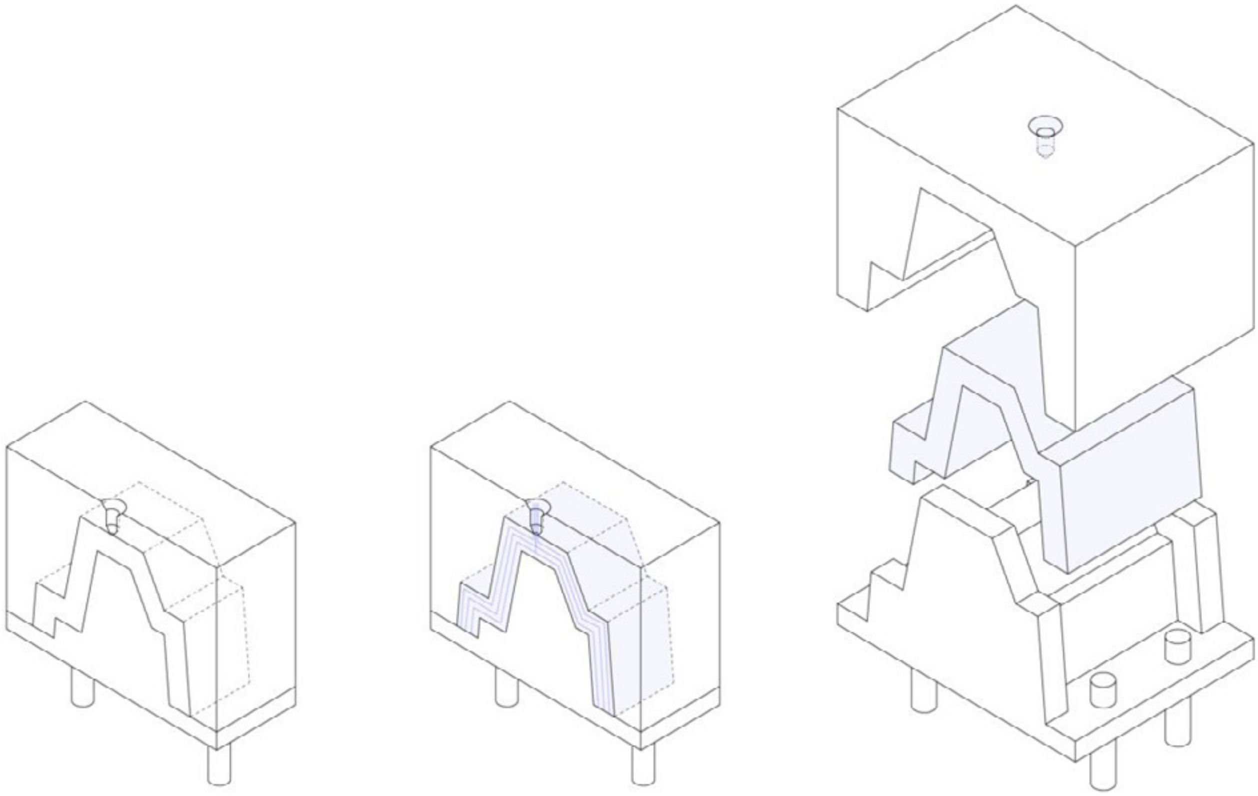
Part 1

# The Basics

What is a Injection molding? How does it work? What is it used for?
In this section, we answer these questions and show you common examples of injection molded parts to help you familiarize with the basic mechanics & applications of the technology.

# What is Injection molding?

Injection molding is a formative manufacturing technology: to create a part, plastic is first melted and then injected into the cavity of a mold. When the material cools, it solidifies and takes the geometry (form) of the mold. The part is then ejected and the process starts over.



This is a fundamentally different way of manufacturing compared to additive (3D printing) or subtractive (CNC machining) technologies. The flow and solidification of the material during injection have a significant impact on the key design restrictions for this technology - more on this in below.

Injection molding is widely used today for both consumer products and engineering applications. Almost every plastic item around you was manufactured using Injection molding. This is due to the ability of the technology to produce identical parts at very high volumes (typically, 1,000 to 100,000+ units) at a very low cost per part (typically, at $1-5 per unit).

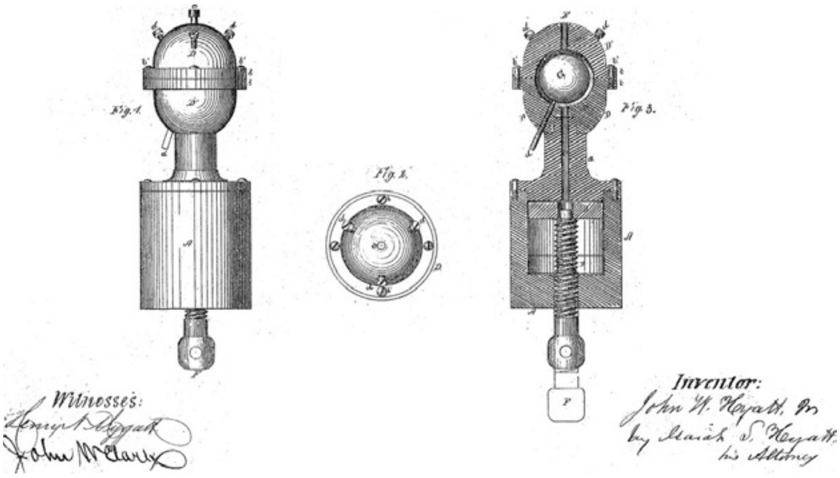
Compared to other technologies though, the start-up costs of Injection molding are relatively high, mainly due to the need for custom tooling. A mold can costs anywhere between $3,000 and $100,000+, depending on its complexity, material (aluminum or steel) and accuracy (prototype, pilot-run or full-scale production mold).

All thermoplastic materials can be Injection molded. Some types of silicone and other thermoset resins are also compatible with the injection molding process. The most commonly used materials in Injection molding are:

- Polypropylene (PP): ~38% of global production
- ABS: ~27% of global production
- Polyethylene (PE): ~15% of global production
- Polystyrene (PS): ~8% of global production

Even if we take into account all other possible manufacturing technologies, Injection molding with these four materials alone accounts for more than 40% of all plastic parts produced globally every year!

# A brief history of Injection molding

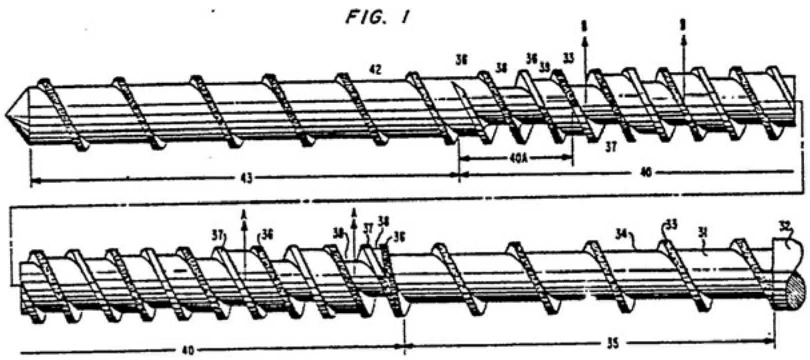



## Plastics replace ivory

In 1869, John Wesley Hyatt invented celluloid, the first practical artificial plastic intended to replace ivory for the production of... billiard balls! Early Injection molding machines used a barrel to heat up the plastic and a plunger to inject it to the mold.

## The revolutionary invention

In the mid-1950s, the invention of the reciprocating screw single-handedly revolutionized the plastics industry. The reciprocating screw solved key issues with uneven heating of the plastic that previous systems faced, and opened up new horizons for the mass production of plastic parts.



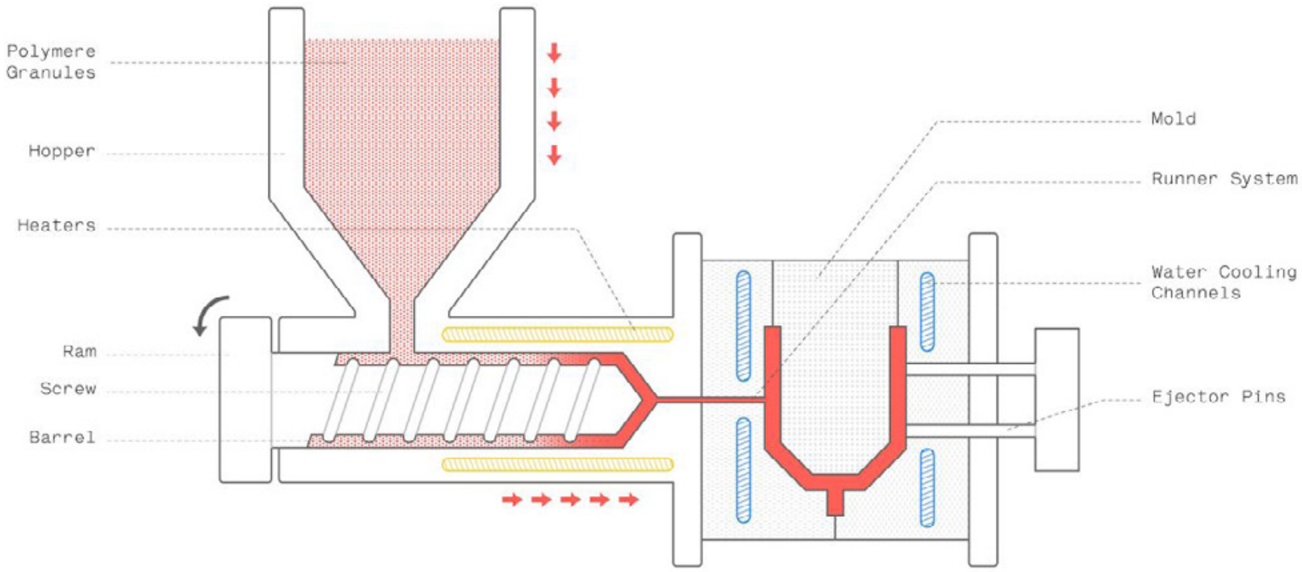

## Injection molding today

Today, injection molding is a $300 billion market. 5+ million metric tons of plastic parts are produced with injection molding globally each year. Recently, the demand of biodegradable materials is increasing for environmental reasons.

# Injection molding machines: How do they work?

An injection molding machine consists of three main parts: the injection unit, the mold - the heart of the whole process - and the clamping/ejector unit. In this section, we examine the purpose of each of these systems and how their basic operation mechanics affect the end-result of the Injection molding process.

# The injection unit

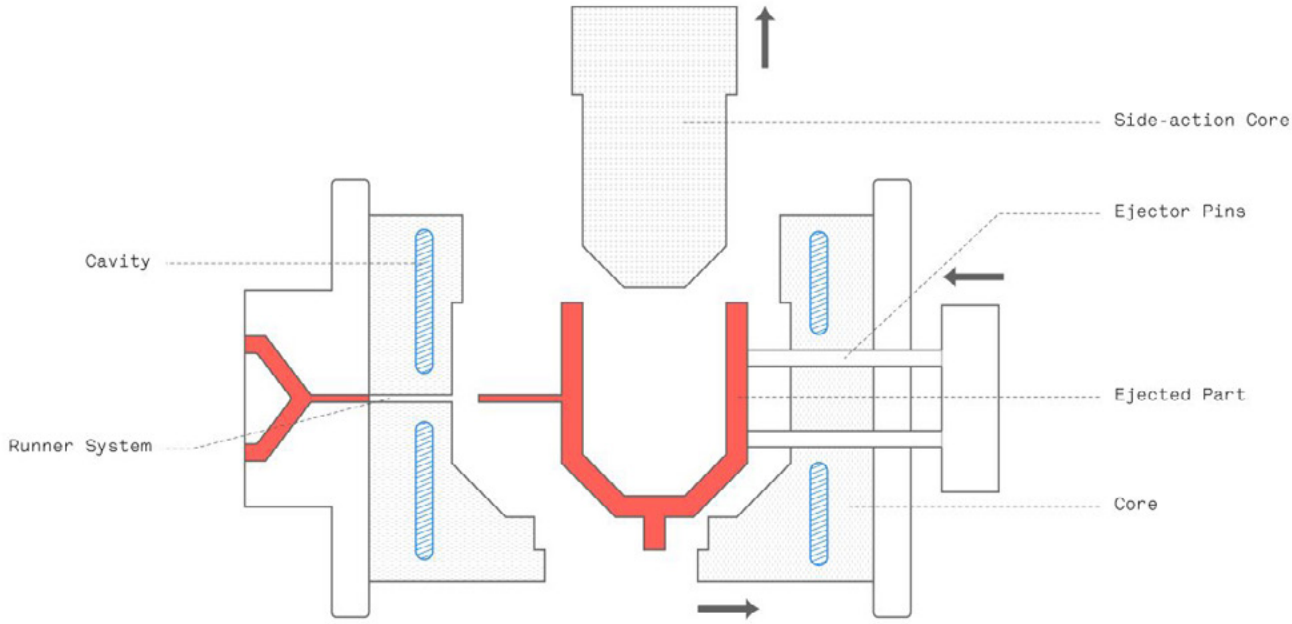



The purpose of the injection unit is to melt the raw plastic and guide it into the mold. It consists of the hopper, the barrel, and the reciprocating screw. The raw plastic material comes as pellets. In the hopper, the pellets can be mixed with pigment or other additives (for example, glass fibers). This way, the color and physical properties of the molded parts can be tailored to meet the specific needs of each application. Next, the material is fed into the barrel, which contains the reciprocating screw.

The screw performs two tasks: it carries the pellets towards the mold and at the same time it compresses them. In fact, the shear forces caused by the movement of the screw produces 60% to 90% of the heat needed to melt the plastic pellets. The rest is provided by the heater bands that are wrapped around the barrel.

Once enough melted plastic is in front of the screw, the ram plunges forward and injects the material into the empty cavity of the mold (like a syringe). This whole process happens in continous fashion, so filling the mold only takes a few seconds.

# The Anatomy of the Mold




The simplest mold - the straight-pull mold - consists of two halves: the cavity and the core. For more complex parts with 'undercuts, side-action cores can also be used that slide in and out of the part on an angle - more on side-action cores & undercuts in a next section.

The core and the cavity have different functions. The core is the half of the mold that is closer to the injection system. It forms the cosmetic side of the part (A-side) that requires a good visual appearance. The cavity is the back half of the mold and it forms the "hidden" functional side (B-side) that includes all structural elements of the part (ribs, bosses etc).

The molds are usually CNC machined from aluminum (for 1,000 to 5,000 units) or tool steel (for 100,000+ units). For low-production runs (< 100 units) the molds can even be 3D printed to expedite lead times.

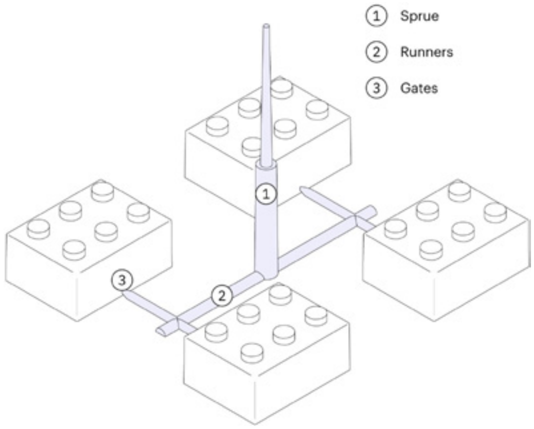
A part from the "negative" of the part, a mold has other features that support the injection process. For example, molds often include cooling channels that accelerate the solidification process and vents that help evacuate the air from the empty mold.

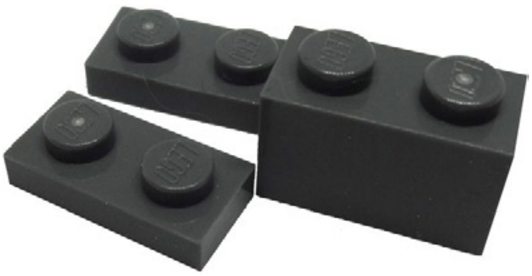
Interesting fact: About 50% of the typical Injection molding cycle is dedicate to part cooling and solidification. Minimizing the thickness of a design is key to speed up this step & cuts costs.

## The Runner System

The melted plastic enters into the mold through the runner system. The runner system usually consists of three main sections: the sprue (the main channel), the runners (the guiding channels) and the gates (the entry points).

Different gates types are suitable for different applications. The illustration shows an edge gate, while the bricks below were manufactured using a hot tip gate - more on gate types here. The runner system is cut off from the part after ejection. This is the only material waste in Injection



## The Vestige

In the point where the runner system connected with the part, a small imperfection is usually visible, called the vestige. If the presence of the vestige is not desirable for asthetic purposes, then in can also be “hidden” in the functional B-side of the part.

# The Clamping & Ejection System

On the far side of an Injection molding machine, there is the clamping system. The clamping system has a dual purpose: it keeps the two parts of the mold tightly shut during injection and it pushes the part out of the mold after it opens. After the part is ejected, it falls on a conveyor belt or a bucket for storage and the cycle starts over again.

Alignment of the different moving parts of the mold is never perfect though. This causes the creation of two common imperfections that are visible on almost every Injection molded part:

- Parting lines which are visible on the side of a part where the two halves of the mold meet. They are caused by tiny misalignments and the slightly rounded edges of the mold.
- Ejector (or witness) marks which are visible on the hidden B-side of the part. They are created because the ejector pins are slightly protruding above or indented below the surface of the mold.

The image below shows the mold used to manufacture both sides of the casing for a remote controller. Quick quiz: try to locate the core (A-side), the cavity (B-side), the runner system, the ejector pins, the side-action core and the air vents on this mold.

# Benefits & Limitations of Injection molding

Injection molding is an established manufacturing technology with a long history, but it is constantly being refined and improved with new technological advancements. Below is a quick rundown of the key advantages and disadvantages of Injection molding to help you understand whether it is the right solution for your application.

# Benefits of Injection molding

## High-volume manufacturing of plastics

Injection Molding is the most cost-competitive technology for manufacturing high volumes of identical plastic parts. Once the mold is created and the machine is set up, additional parts can be manufactured very fast and at a very low cost.

The recommended minimum production volume for injection molding is 500 units. At this point economies of scale start to kick-in and the relatively high initial costs of tooling have a less prominent effect on the unit price.

## Wide range of materials

Almost every thermoplastic material (and some thermosets and silicones) can be injection molded. This gives a very wide range of available materials with diverse physical properties to design with.

Parts produced with Injection molding have very good physical properties. Their properties can be tailored by using additives (for example, glass fibres) or by mixing together different pellets (for instance, PC/ABS blends) to achieve the desired level of strength, stiffness or impact resistance.

## Very high productivity

The typical injection molding cycle last 15 to 60 seconds, depending on the size of the part and the complexity of the mold. In comparison, CNC machining or 3D printing might require minutes to hours to produce the same geometry. Also, a single mold can accomodate multiple parts, further increasing the production capabillities of this manufacturing process.

This means that hundreds (or even thousands) of identical parts can be produced every single hour.

## Great repeatability & tolerances

The Injection molding process is highly repeatable and the produced parts are essentially identical. Some wear of course occurs to the mold over time, but a typical pilot-run aluminum mold will last 5,000 to 10,000 cycles, while full-scale production molds from tool steel can stand 100.000+ cycles.

Typically, Injection molding will produce parts with tolerances of ± 0.500 mm (0.020"). Tighter tolerance down to ± 0.125 mm (0.005") are also feasible in certain circumstances. This level of accuracy is enough for most applications and comparable to both CNC machining and 3D printing.

## Excellent visual appearance

A key strength of Injection molding is that it can produce finished products that need little to no extra finishing. The surfaces of the mold can be polished to a very high degree to create mirror-like parts. Or they can be bead blasted to create textured surfaces. The SPI standards dictate the level of finishing that can be achieved.

Get the finishing/material compatibility recommendations →

# Limitations of Injection molding

## High start-up costs for tooling

The main economic restriction in Injection molding in the high cost of tooling. Since a custom mold has to be made for each geometry, the start-up costs are very high. These are mainly related to the design and manufacturing of the mold and typically varies between $5,000 and $100,000.

For this reason, injection molding is only economically viable for productions larger than 500 units.

## Design changes are costly

After a mold is manufactured, it is very expensive to modify it: design changes usually require the manufacture of a new mold from scratch. For this reason, correctly designing a part for Injection molding is very important.

In Part 2, we list the most important design consideration you have to keep in mind while designing for Injection molding. It Part 5, we will also see how you can mitigate the risk by creating physical prototypes of your parts.

## Longer lead times than other technologies

The typical turnaround for injection molding varies between 6 to 10 weeks - 4 to 6 weeks for manufacturing the mold, plus 2 to 4 more weeks for production and shipping. If design changes are required to the model, something quite common, the turnaround time increases accordingly. In comparison, parts made in a desktop 3D printer can be ready for delivery overnight, while industrial 3D printing systems have a typical lead time of 3-5 days. CNC machined parts are typically delivered within 10 days or as fast as 5 days.

# Examples of Injection molding products

If you look around you right now, you will see at least a few products that were manufactured with Injection molding. You are probably looking at one right now actually: the casing of the device you are using to read this guide. To recognize them, look out for these three things: a parting line, witness marks on the hidden side and a relatively uniform wall thickness throughout the part.

We collected here some examples of common Injection molding products to help get a better understanding of what can be achieved with this manufacturing process.

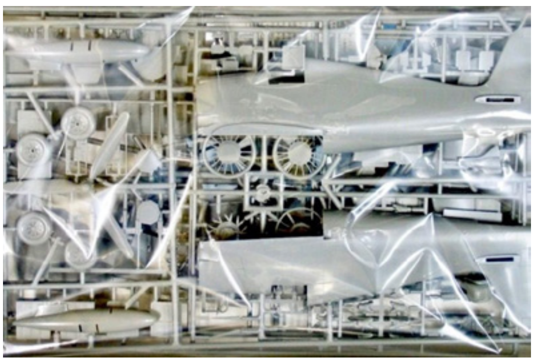

## Toys

Lego bricks are one of the most recognisable examples of Injection molded parts. They are manufactured using molds like the one in the picture, which produced 120 million lego bricks (that is 15 million cycles) before it was taken out of commission.The material used for Lego bricks is ABS due to its high impact resistance and excellent moldabillity. Every single brick has been designed to perfection achieving tolerances down to 10 micro meters (or a tenth of a human hair). This is partly achieved by using the best design practices we will examine in the next section (uniform wall thickness, draft angles, ribs, embossed text etc).

## Packaging

Many plastic packaging products are Injection molded. In fact, packaging is the largest market for Injection molding. For example, bottle caps are Injection molded from Polypropylene. Polypropylene (PP) has excellent chemical resistance and is suitable to come in contact with food products.

On bottle caps, you can also see all common Injection molding imperfections (parting line, ejector marks etc) and common design features (ribs, stripping undercuts etc).



## Miniatures

Model airplanes are another common example of Injection molded parts. The material used here is mostly Polystyrene (PS), for its low cost and ease of molding. What is interesting with model airplane kits is that they come with the runner system still attached. So, you can see the path the melted plastic followed to fill the empty mold.

## Automotive

Almost every plastic component in the interior of a car was Injection molded. The three most common injection molding materials used in the automotive industry are Polypropylene (PP) for non-critical parts, PVC for its good weather resistance and ABS for its high impact strength.

More than half of the plastic parts of a car are made from one of these materials, including the bumpers, the interior body parts and the dashboards

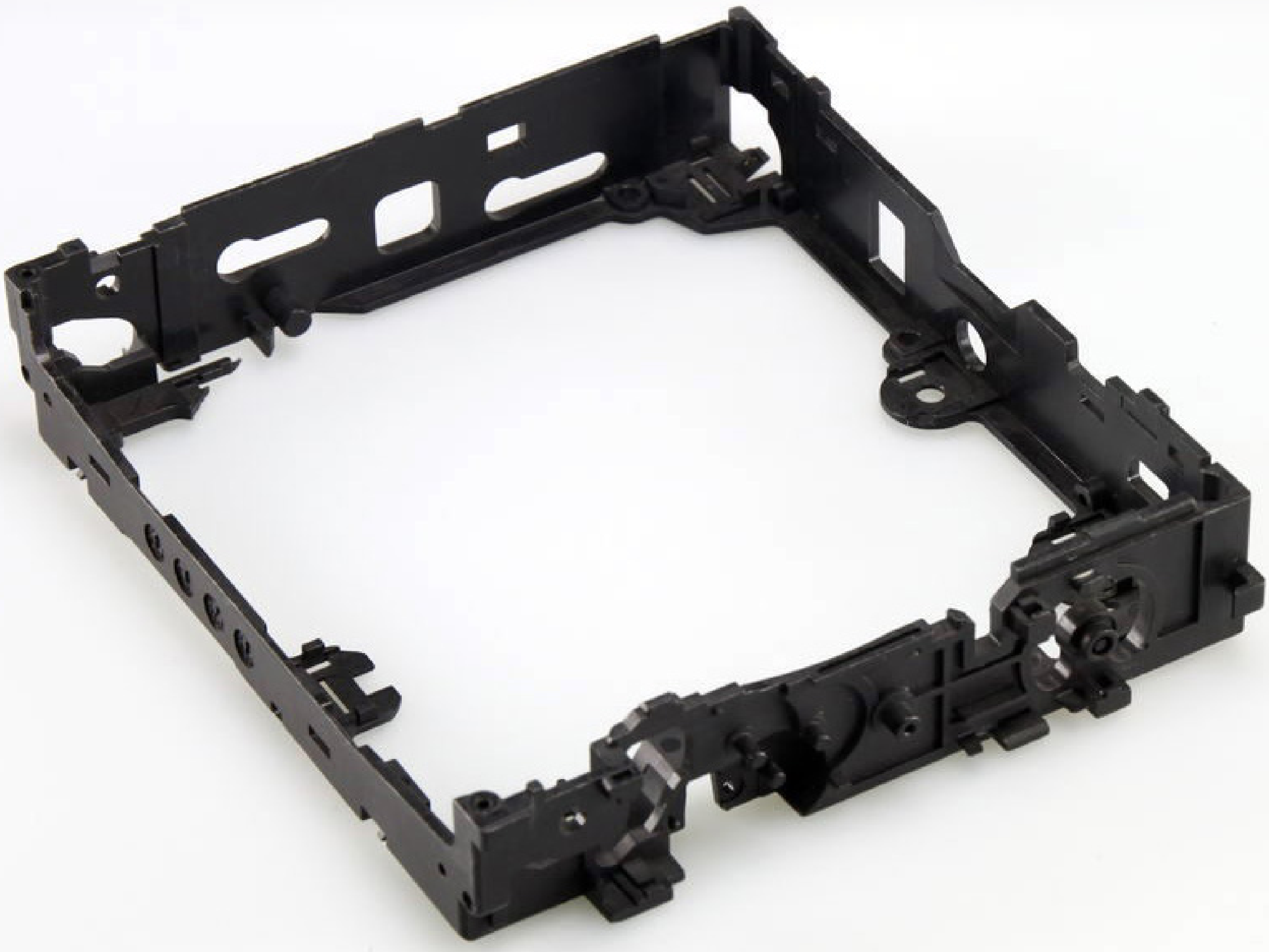
## Electrical

The enclosures of almost every mass-produced consumer electronic device was Injection molded. ABS and polystyrene (HDPE) are prefered here for their excellent impact resistance and good electrical insulation.

## Healthcare

Many sterilizable and biocompatible materials are available for Injection molding. Medical grade silicone is one of the more popular materials in the medical industry. Silicone is a thermoset though, so special machinery and process control are required, increasing the cost.

For applications with less strict requirements other materials, like ABS, Polypropylene (PP) and Polyethylene (PE), are more common.



Part 2

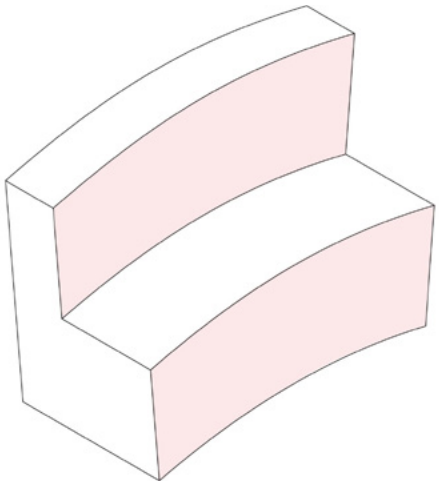
# Design for Injection molding

In this section you will learn how to optimize your designs for Injection molding. Use the following guidelines to save time and reduce failures and learn how to create features that maximize the functionallity of your designs.

# Common injection molding defects

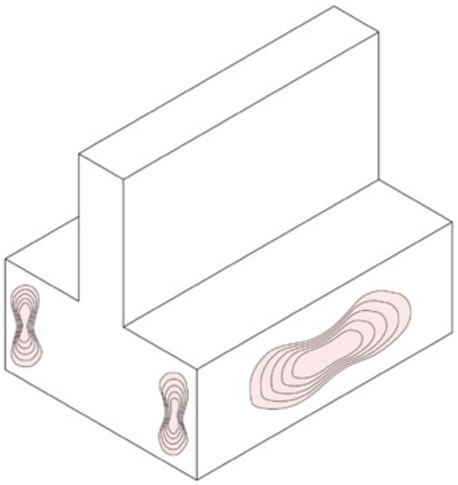
Most defects in Injection molding are related with either the flow of the melted material or its non-uniform cooling rate during solidification.

Here is a list of defects than an engineer should keep in mind while designing a part for Injection molding. In the next section, we will see how you can avoid each of them by following good design practices.



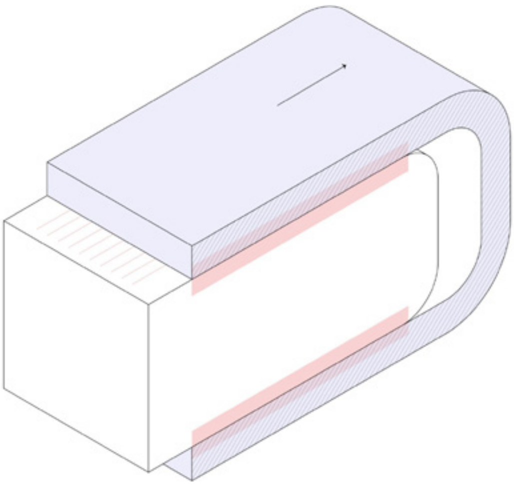
## Warping

When certain sections cool (and as a result shrink) faster than others, then the part can permanently bend due to internal stresses. Parts with non-constant wall thickness are most prone to warping.



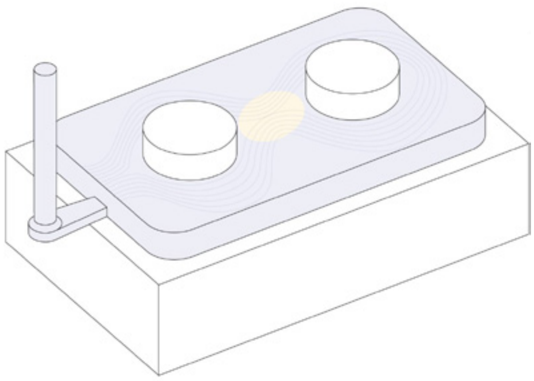
## Sink marks

When interior of a part solidifies before its surface, a small recess in an otherwise flat surface may appear, called a sink mark. Parts with thick walls or poorly designed ribs are most prone to sinking.



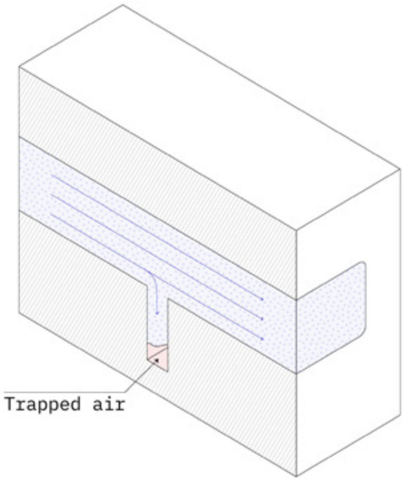
## Drag marks

As the plastic shrinks, it applies pressure on the mold. During ejection, the walls of the part will slide and scrape against the mold, which can result to drag marks. Parts with vertical walls (and no draft angle) are most prone to drag marks.



## Knit lines

When two flows meet, small hair-like discolorations may develop. These knit lines affect the parts aesthetics, but also they generally decrease the strength of the part. Parts with abrupt geometry changes or holes are more prone to knit lines.

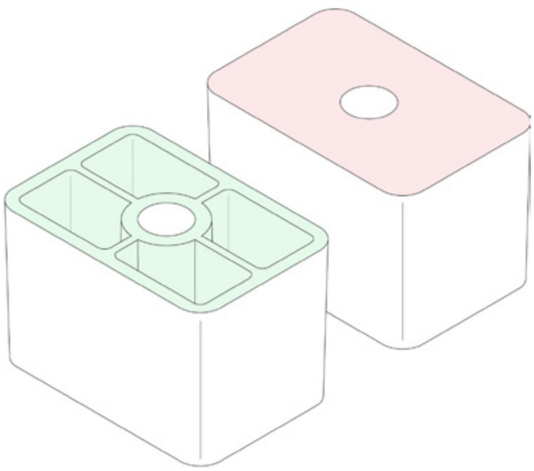



## Short shots

Trapped air in the mold can inhibit the flow of the material during injection, resulting in an incomplete part. Good design can improve the flowability of the melted plastic. Parts with very thin walls or poorly designed ribs are more prone to short shots.

# Design Rules for Injection Molding

Let's see how these process restriction can be translated into actionable design guidelines. In the following sections, we summarize the most important design rules to follow when designing parts for Injection molding, as well as tips on how to design the most common features found in Injection molded parts correctly.



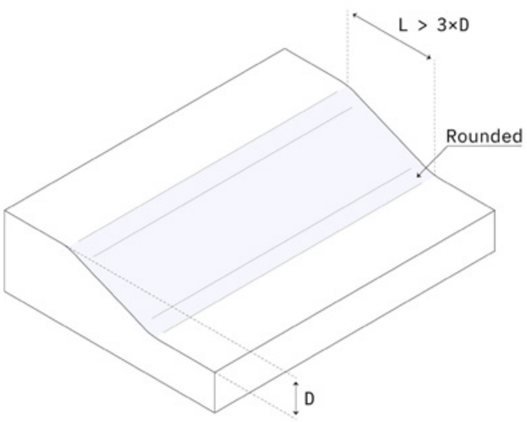
## Use a constant wall thickness

Recommended thickness: 1 mm and 3 mm

Always design parts with the smallest possible (and constant) wall thickness, to avoid warping and sinking.

If thicker sections are required, hollow them out and use ribs to add stiffness instead. Keep in mind that each 10% increase in wall thickness provides approximately a 30% increase in stiffness.

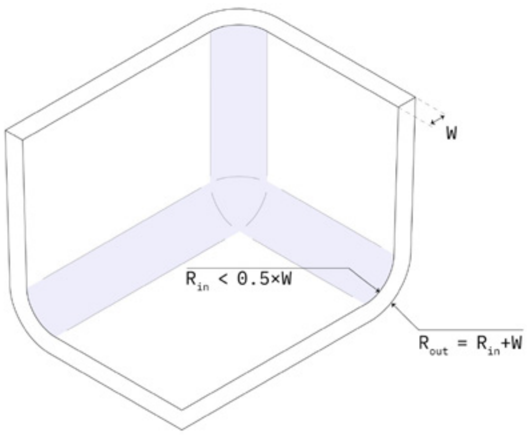
See material specific wall thickness recommendations →




## Add smooth transitions

Recommended: 3 × wall thickness difference

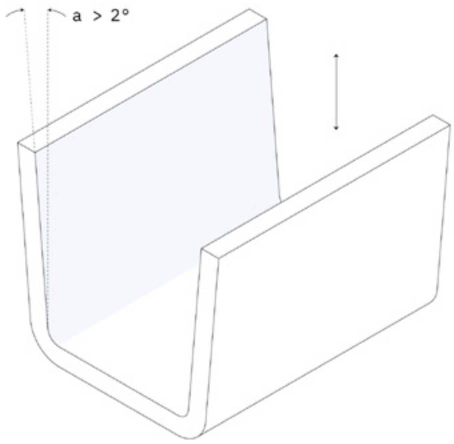
Sometimes sections with different wall thickness cannot be avoided. In these cases, use a chamfer or fillet to make the transition as smooth as possible.Similarly, the base of vertical features (like ribs, bosses, snap-fits) must also always be rounded.




## Round all edges

Internal edges: > 0.5 × wall thickness
External edges: internal fillet + wall thickness

The constant wall thickness rule must also be applied to the corners of the part. Add a fillet with a radius that is as large as possible to all internal and external edges.




## Add draft angles

Recommended minimum: > 2°

Add a draft to all vertical walls to make the ejection of the part easier and avoid drag marks. If they serve a functional purpose, external walls may be left undrafted (see Lego bricks). Increase the draft angle above the recommended in these cases:

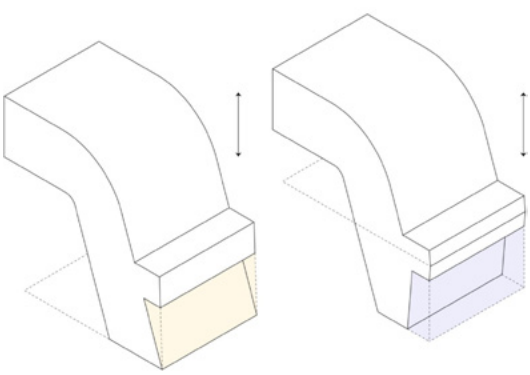
- For parts taller than 50 mm, increase the draft by 1° for every 25 mm
- For parts with a textured finish, increase the draft by an extra 1°-2°

# Dealing with undercuts

An important aspect to consider while designing parts for Injection molding are undercuts.

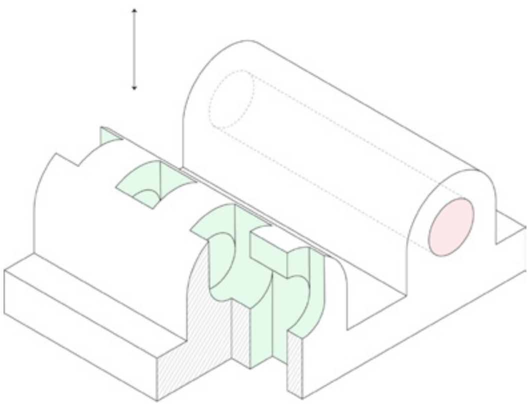
`Undercuts in Injection molding are part features that cannot be manufactured with a simple two-part mold, because material is in the way while the mold opens or during ejection.`

The teeth of a thread or the hook of a snap-fit joint are examples of undercuts. Here are some simple solutions to help you deal with undercuts:



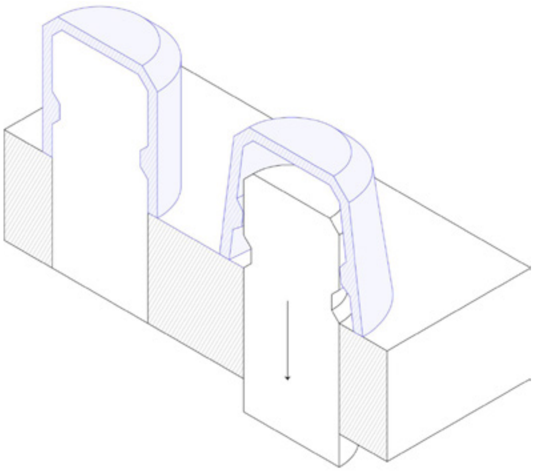
## Moving the parting line

The simplest way to deal with an undercut is to move the parting line of the mold to intersect with it. This solution is suitable for many designs with undercuts on an external surface. Don't forget to adjust the draft angles accordingly.



## Using a shut-off

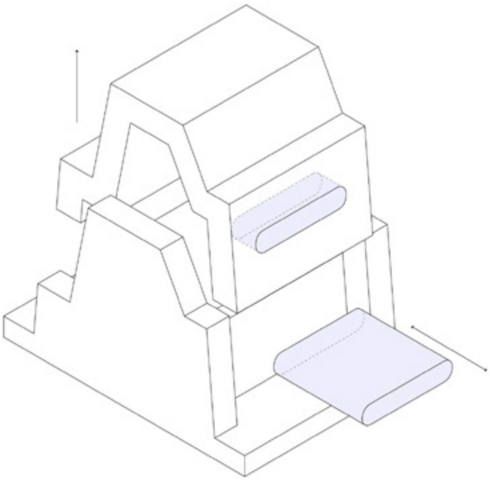
Another way to deal with undercuts is to remove material from under or above the problematic area. This way the undercut is eliminated as the whole part can be directly supported by the mold. Shut-offs are a useful trick to deal with undercuts on internal regions of the part (for snap-fits) or on the sides of the part (for holes or handles).



## Stripping undercuts

If the part is flexible enough, then deforming over the mold during ejection is an option. Stripping undercuts are used for internal features, such as the threads of bottle caps. Use these guidelines to design stripping undercuts:

- Select a flexible material - such as PP, PE or Nylon (PA)
- The height of the undercut should be 5% the diameter of the hole
- Use a lead angle of 30° to 45°



## Side-action cores

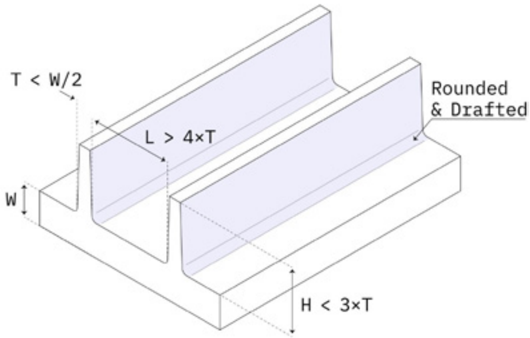
When none of the above solutions are viable, then cores can be used that slide out of the part from the side before it is ejected. Side-action cores should be used sparingly as they add complexity and increase the overall cost of a mold by 15% to 30%. Follow these guidelines when designing a side-action core:

- The core must move in a direction parallel to the parting line
- Draft angles must be added as usual

# Common design features

We list below practical guidelines on how to design the most common features encountered in Injection molded parts. Use them to improve the functionality of your designs, while still complying with the basic design rules.

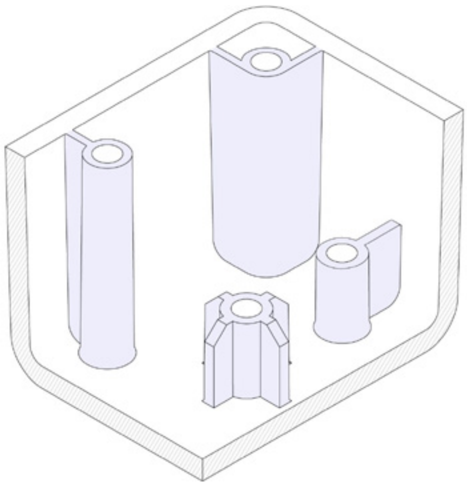
Read the full design guidelines for Injection molding →




## Ribs

When the even the maximum recommended wall thickness is not enough to meet the functional requirements of a part, ribs can be used to improve its stiffness. When designing ribs:

- Use a thickness equal to 0.5 × main wall thickness
- Define a height smaller than 3 × rib thickness
- Use a base fillet with radius greater then ¼ × rib thickness
- Add a draft angle of at least 0.25° - 0.5°
- Add a min. distance between ribs & walls of 4 × rib thickness



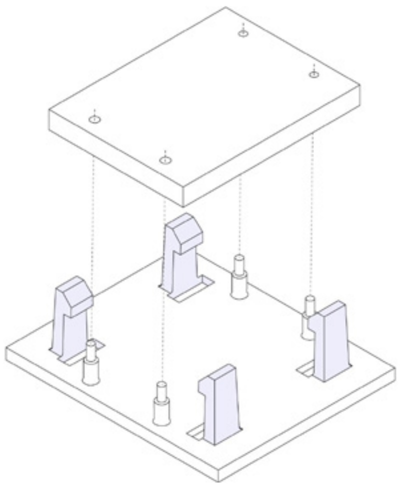
## Bosses

Bosses are used as points of attachment or fastening (in conjuction with self-tapping screws or threaded inserts). Think of bosses as circular ribs - the same general design guideline apply. Consider also the following:

- Avoid designing bosses that merge into main walls
- Support bosses with ribs or connect them to a main wall

For bosses with inserts:

- Use an outer diameter equal to 2 × the insert's nominal size

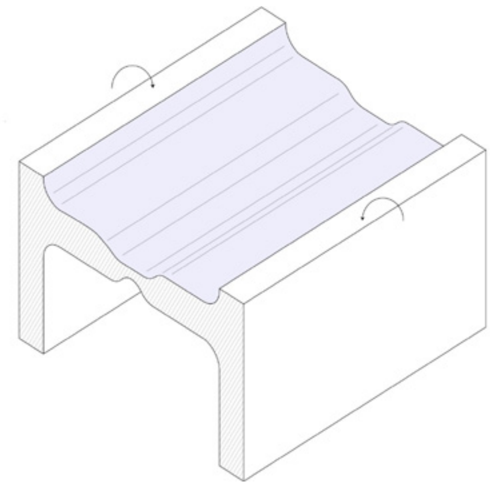


## Snap-fit joints

Snap-fits are an economical and rapid way of joining two parts without fasteners or tools. When designing snap-fits for Injection molding:

- Add a draft to the side-walls of the snap-fit
- Use a thickness of 0.5 × main wall thickness
- Adjust the width & length to control the deflection & force
- Think how to deal with the created undercut

For detailed guidelines, refer to this article from MIT.

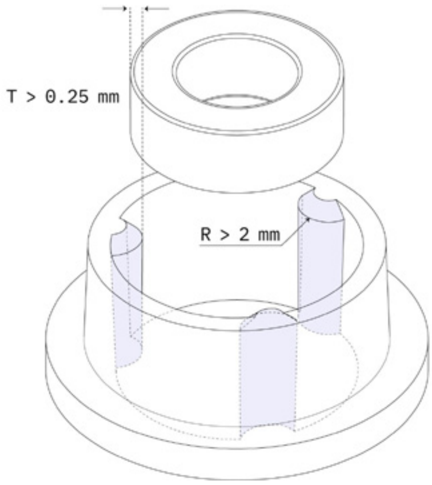


## Living hinges

Living hinges are thin sections of plastic that connect two segments of a part and allow it to flex and bend. Here are some tips to help design a living hinge:

- Select a flexible material (for example PP, PE or Nylon)
- Design hinges with a thickness between 0.20 and 0.35 mm
- Use shoulders with a thickness equal the thickness of the main wall
- Add as large fillets as possible

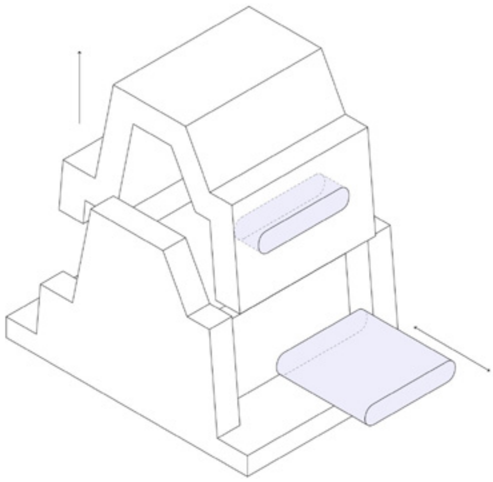
For detailed guidelines, please refer to this MIT guide.




## Crush ribs

Crush ribs deform and create friction between the part and the inserted component, securing it in place. They are a fast and inexpensive method of incorporating bearings or other inserts in your designs. For high end applications, consider using a press fit instead. When designing crush ribs:

- Use three circular ribs with a 2 mm radius
- Add a min. overlap of 0.25 mm between the rib and the insert
- Add draft to the hole but do not draft the ribs



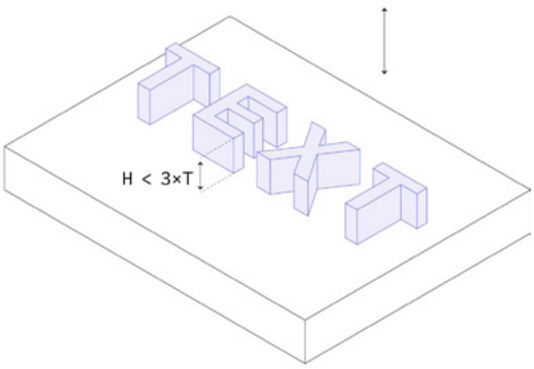
## Threads

Threads can be added directly to the molded part design, but introduce undercuts. Alternatively, threaded inserts can be used. Follow these guidelines when you design parts with threads:

- Add a 0.8 mm relief at the edges of the thread
- Use a thread with a pitch greater than 0.8 mm (32 threads per inch)
- Prefer using a trapezoidal or buttress thread

To deal with the created undercuts:

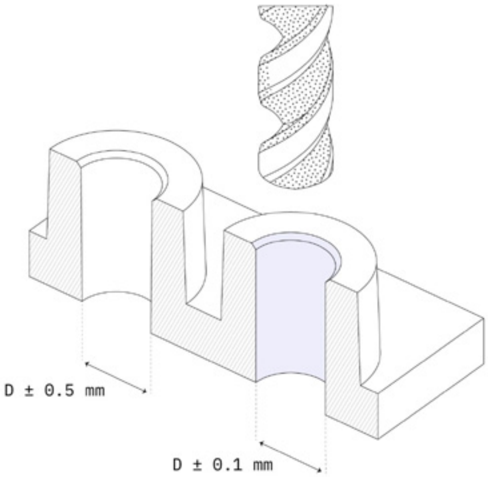
- For internal threads, consider using stripping undercuts
- For external threads, place them along the parting line




## Lettering & symbols

Text, logos and other symbols can be engraved or embossed on the surface of Injection molded parts. Here are some tips while adding text:

- Prefer embossing instead of engraving
- Align the text perpendicular to the parting line
- Use a height (or depth) greater than 0.5 mm
- Use a font with uniform letter thickness
- The font size should be at least 20 points




## Tolerances

Injection molding typically produces parts with tolerances of ± 0.500 mm (0.020”). Tighter tolerances are feasible in certain circumstances (down to ± 0.125 mm - and even ± 0.025 mm), but they increase the cost drastically.

For small production runs (< 10,000 units), consider using a secondary operation (such as drilling) to improve accuracy. This ensures the correct interference of the part with other components or inserts (for example, when using press fits).

Part 3

# Injection molding materials

Injection molding is compatible with a very wide range of plastics. In this section, you will learn more about the key characteristics of the most popular materials. We will also visit the standard surface finishes that can be applied to Injection molded parts.

# Injection molding materials

All thermoplastics can be injection molded. Some thermosets and liquid silicones are also compatible with the injection molding process. They can be also reinforced with fibers, rubber particles, minerals or flame retardant agents to modify their physical properties. For example Fiberglass can be mixed with the pellets at ratios of 10%, 15% or 30% resulting in parts with higher stiffness.

## Polypropylene (PP)

The most common Injection molding plastic. Excellent chemical resistance. Food-safe grades available. Not suitable for mechanical applications.

## ABS

Common thermoplastic with high impact resistance, low-cost & low density. Vulnerable to solvents.

## Polyethylene (PE)

Lightweight thermoplastic with good impact strength & weather resistance. Suitable for outdoor applications.

## Polystyrene (PS)

The Injection molding plastic with the lowest cost. Food-safe grades available. Not suitable for mechanical applications.

## Polyurethane (PU)

Thermoplastic with high impact strength and good mechanical properties & hardness. Suitable for molding parts with thick walls.

## Nylon (PA 6)

Engineering thermoplastic with excellent mechanical properties and high chemical & abrasion resistance. Susceptible to moisture.

## Polycarbonate (PC)

The plastic with the highest impact strength. High thermal resistance, weather resistance & toughness. Can be colored or transparent.

## PC/ABS

Blend of two thermoplastics resulting in high impact strength, excellent thermal stability, and high stiffness. Vulnerable to solvents.

## POM (Acetal/Delrin)

Engineering thermoplastic with high strength, stiffness & moisture resistance and self-lubricating properties. Relatively prone to warping.

## PEEK

High-performance engineering thermoplastic with excellent strength and thermal & chemical resistance. Used to replace metal parts.

## Silicone rubber

Thermoset with excellent heat & chemical resistance and customizable shore hardness. Food-safe and medical grade available.

## Surface finishes & SPI standards

Injection molded parts are not usually post-processed, but the mold itself can be finished to various degrees. This way aesthetic needs (for example, a mirror-like or matte surface) or technical requirements (for example, specific surface roughness or tolerances) can be achieved.

The Society of Plastics Industry (SPI) several standard finishing procedures that result in different part surface finishes.

| Finish | Description | Application |
|---|---|---|
| Glossy finish<br>SPI standard: A-1, A-2, A-3 | The mold is smoothed and then polished with a diamond buff, resulting in parts with a mirror-like finish. | Suitable for parts that need the smoothest surface finish for cosmetic or functional purposes (Ra < 0.10 μm) |
| Semi-gloss finish<br>SPI standard: B-1, B-2, B-3 | The mold is smoothed with fine grit sandpaper, resulting in parts with a fine surface finish. | Suitable for parts that require a good visual appearance, but not a high glossy look. |
| Matte finish<br>SPI standard: C-1, C-2, C-3 | The mold is smoothed using fine stone powder, removing all machining marks. | Suitable for parts with low aesthetic requirements, but when machining marks are not acceptable. |
| Textured finish<br>SPI standard: D-1, D-2, D-3 | The mold is first smoothed with fine stone powder and then sandblasted, resulting in a textured surface. | Suitable for parts that require a satin or matte textured surface finish. |
| As-machined finish | The mold is finished to the machinist's discretion. Tool marks may be visible. | Suitable for non-cosmetic, industrial parts or hidden components. |

See a detailed description of the SPI standards & their compatibility with each material →

Part 4

# Cost reduction tips

Learn more about the main cost drivers in Injection molding and three actionable design tips that will help reduce costs you keep your project on budget.

## Cost drivers in Injection molding

The main cost drivers in Injection molding are:

- Tooling costs determined by the total cost of designing and machining the mold
- Material costs determined by the volume of the material used and its price per kilogram
- Production costs determined by the total time the Injection molding machine is used

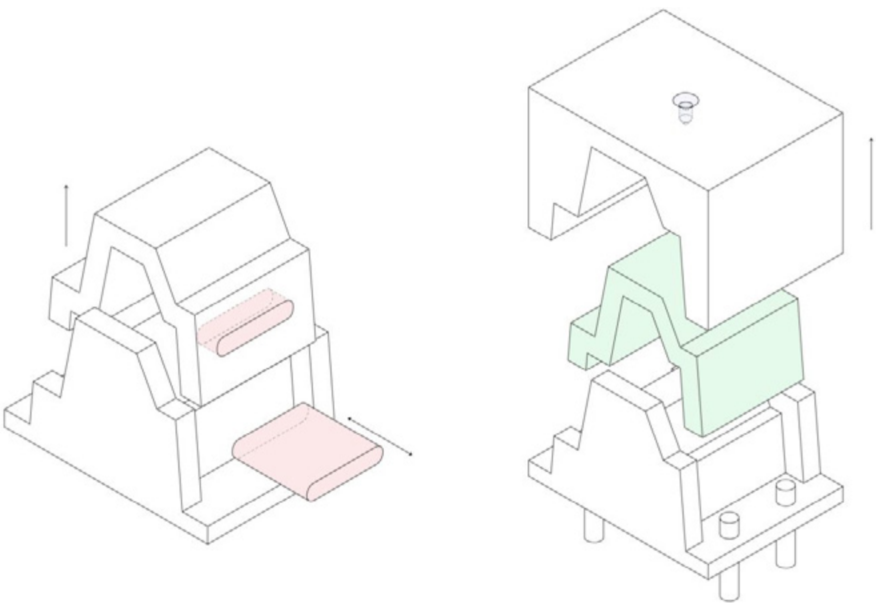
Tooling costs are constant (starting at $3,000 to $5,000) and independent of the total number of manufactured parts, while the material and production costs are dependent on the production volume.

For smaller productions (1,000 to 10,000 units), the cost of tooling has the greatest impact on the overall cost (approximately 50-70%). So, it is worthwhile to alter your design accordingly to simplify the process of manufacturing of the mold (and thus its cost).

For larger volumes to full-scale production (10,000 to 100,000+ units), the contribution of the tooling costs to the overall cost is overshadowed by the material and production costs. So, your main design efforts should focus on minimizing both the volume part and the time of the molding cycle.

Here we collected some tips to help you minimize the cost of your Injection molded project.

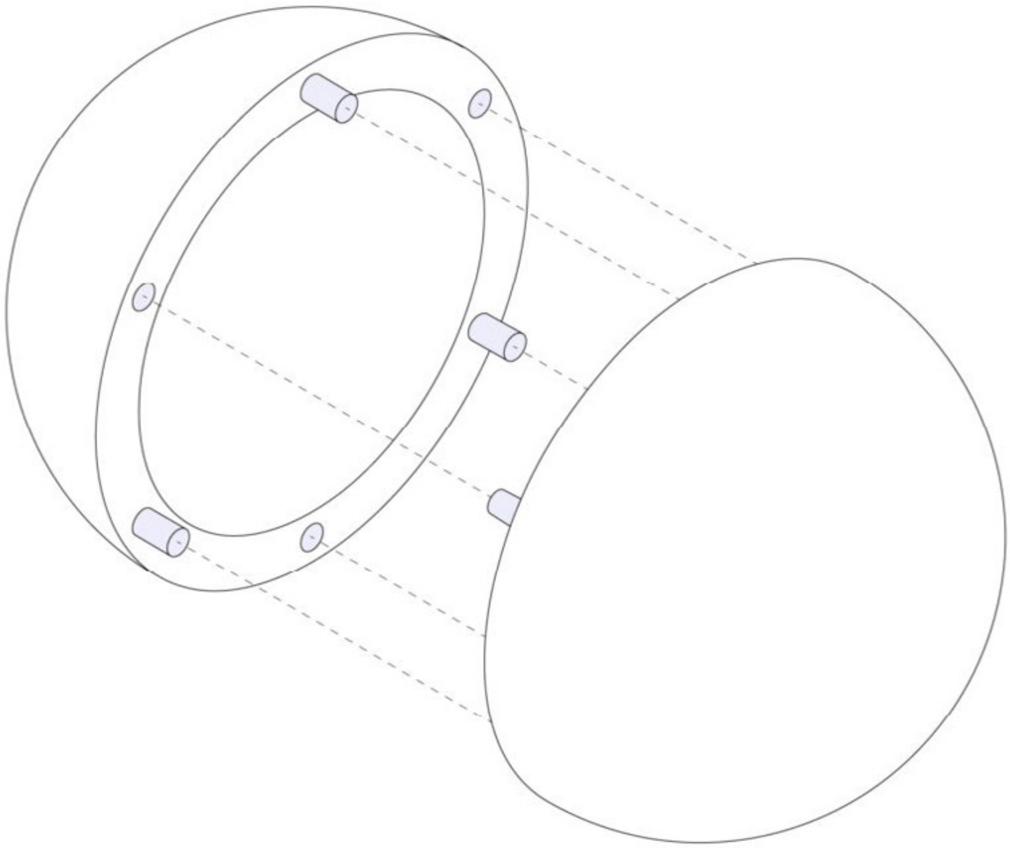
# Tip #1: Stick to the straight-pull mold



Side-action cores and the other in-mold mechanisms can increase the cost of tooling by 15% to 30%. This translates to a minimum additional cost for tooling of approximately $1,000 to $1,500.

In a previous section, we examined ways to deal with undercuts. To keep your production on-budget, avoid using side-action cores and other mechanisms unless absolutely necessary.

# Tip #2:
# Fit multiple parts in one mold



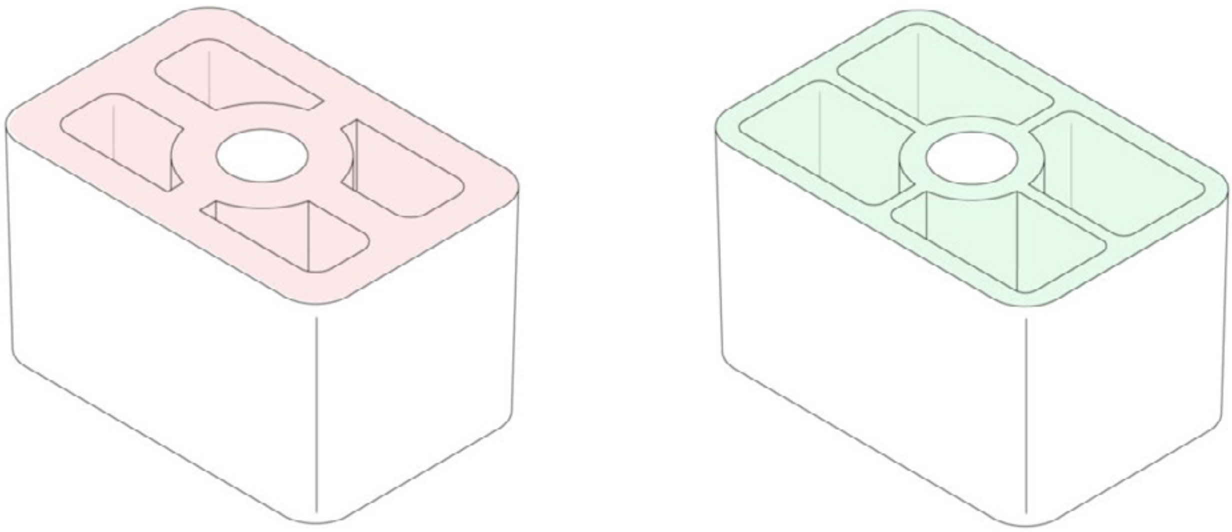
We saw in a previous section, that fitting multiple parts in the same mold is a common practice. Usually, six to eight small identical parts can fit in the same mold, essentially reducing the total production time by about 80%.

Parts with different geometries can also fit in the same mold (remember, the model airplane example). This is a great solution for reducing the overall cost of an assembly. The parts should not

Here's an advanced technique:

In some cases, the main body of two parts of an assembly is the same. With some creative design, you can create interlocks points or hinges at symmetrical locations, essentially mirroring the part. This way the same mold can be used to manufacture both halves, cutting the tooling costs in half.

# Tip #3:
# Minimize the part volume by reducing the wall thickness



Reducing the wall thickness of your part is the best way to minimize the part volume. This way not only less material is used, but also the injection molding cycle is greatly accelerated.

For example, reducing the wall thickness from 3 mm to 2 mm can reduce the cycle time by 50% to 75%.

Thinner walls mean that the mold can be filled faster. More importantly, parts thinner parts cool and solidify much faster. Remember that about half of the Injection molding cycle is spent for the solidification of the part, while the machine is kept idle.

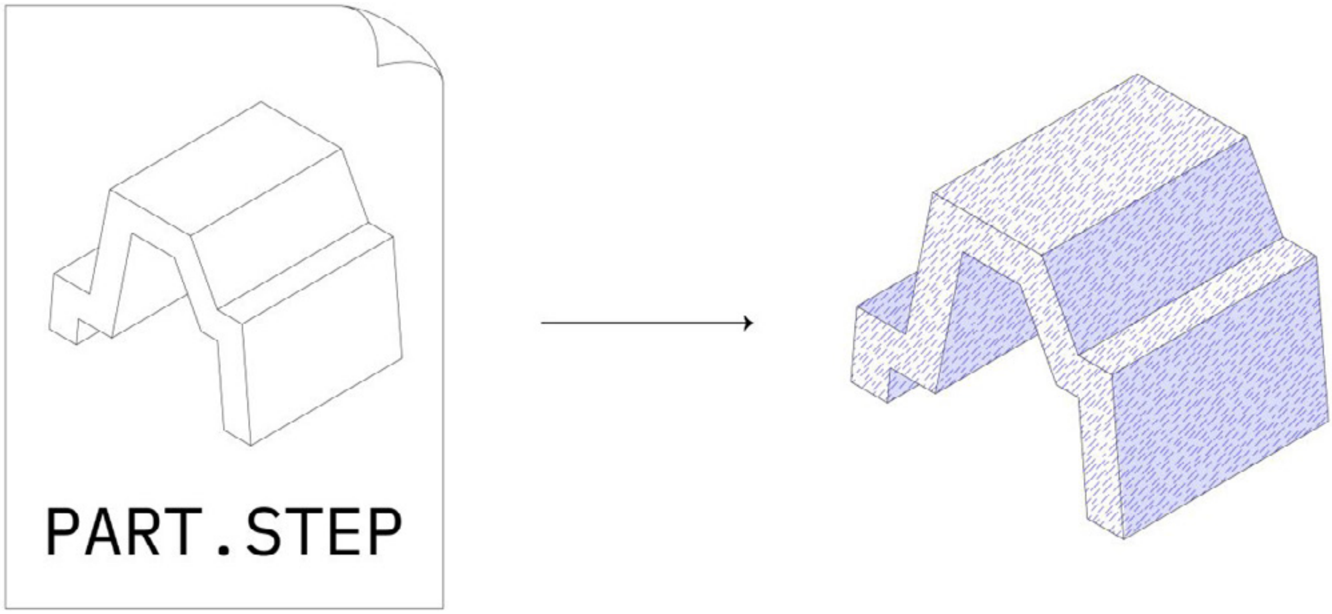
Care must be taken through to not overly reduce the stiffness of the part, downgrading its mechanical performance. Ribs in key locations can be used to increase stiffness.

Part 5

# Start Injection molding

With your design design ready and optimized for injection molding, what's next? In this section we walk you through the steps needed to start manufacturing with injection molding.

# Step 1:
# Start small & prototype fast




Before you commit to any expensive Injection molding tooling, first create and test a functional prototype of your design. This step is essential for launching a successful product. This way design errors can be identified early, while the cost of change is still low.

There are three solutions for prototyping:

1. 3D printing (with SLS, SLA or Material Jetting)
2. CNC machining in plastic
3. Low-run injection molding with 3D printed molds

These three processes can create realistic prototypes for form and function that resemble closely the appearance of the final Injection molding product.

Use the information below as a quick comparison guide to decide which solution is best for your application.

## Prototyping with 3D printing

**Min. quantity:** 1 part
**Typical cost:** $20 - $100 per part
**Lead time:** 2 - 5 days

Learn more about this process →

| | | |
|---|---|---|
| **Pros** | Designs optimized for injection molding can be easily 3D printed | The prototyping solution with the lowest cost & fastest turnaround |
| **Cons** | Not every injection molding material is available for 3D printing | 3D printed parts are 30-50% weaker than injection molded parts |

## Prototyping with CNC Machining

**Min. quantity:** 1 part
**Typical cost:** $100 - $500 per part
**Lead time:** 5 - 10 days

Learn more about this process →

| | | |
|---|---|---|
| **Pros** | Material properties identical to the injection molded parts | Excellent accuracy and finishing |
| **Cons** | Design modifications may be need, as different design restrictions apply | More expensive than 3D printing with longer lead time |

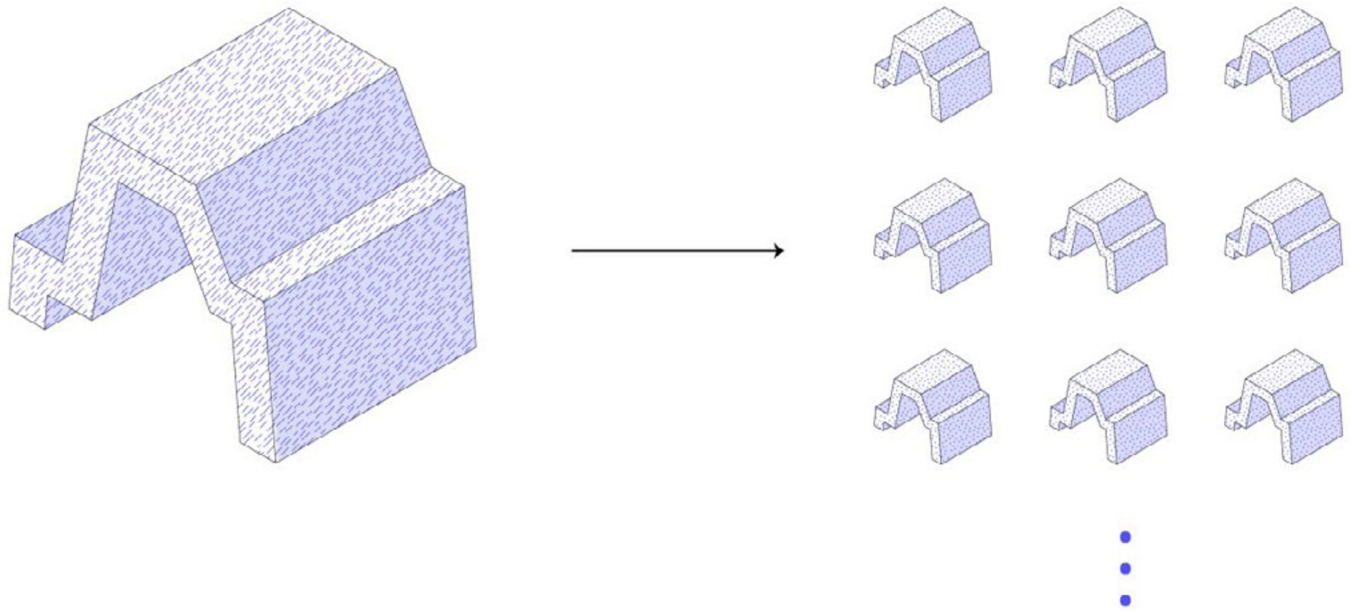
## Prototyping with Low-run Injection molding

**Min. quantity:** 10 - 100 parts
**Typical cost:** $1000 - $4000 per part
**Lead time:** 5 - 10 days

Learn more about this process →

| | | |
|---|---|---|
| **Pros** | The most realistic prototypes with accurate material properties | The actual process and mold design is simulated |
| **Cons** | The prototyping solution with the highest cost | Smaller availability than CNC or 3D printing |

# Step 2:
# Make a “pilot run” (500 - 10,000 parts)



With the design finalized, it time to get started with Injection molding with a small pilot run.

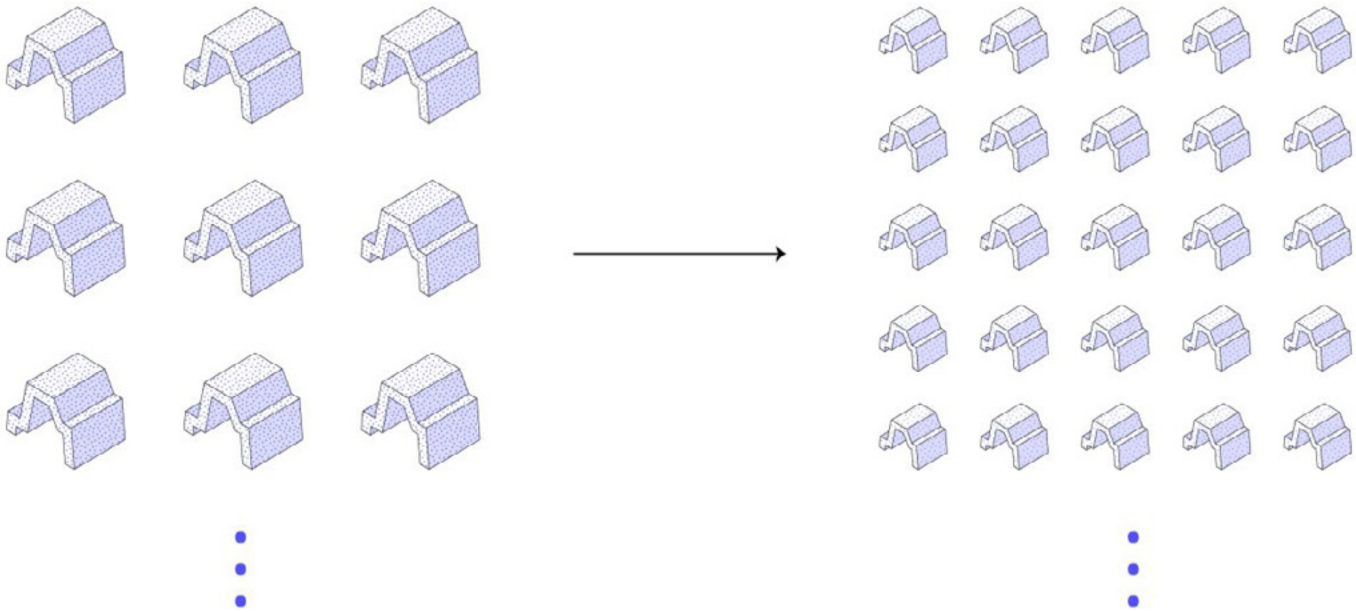
The minimum order volume for injection molding is 500 units. For these quantities, the molds are usually CNC machined from aluminum. Aluminum molds are relatively easy to manufacture and low in cost (starting at about $3,000 to $5,000) but can withstand up to 5,000 - 10,000 injection cycles.

At this stage, the typical cost per part varies between $1 and $5, depending on the geometry of your design and the selected material. The typical lead time for such orders is 6-8 weeks.

`Don’t get confused by the term “pilot run”. If you only require a few thousand parts, then this would be your final production step.`

The parts manufactured with “pilot” aluminum molds have physical properties and accuracy identical to parts manufactured with “full-scale production” tool steel molds.

# Step 3:
# Scale up production (10,000+ parts)



When producing parts massive quantities of identical parts (from 10,000 to 100,000+ units) then special Injection molding tooling is required.

For these volumes, the molds are CNC machined from tool steel and can withstand millions of Injection molding cycles. They are also equipped with advanced features to maximize production speeds, such as hot-tip gates and intricate cooling channels.

The typical unit cost at this stage varies between a few cents to $1 and the typical lead time is 4 to 6 months, due to the complexity of designing and manufacturing the mold.