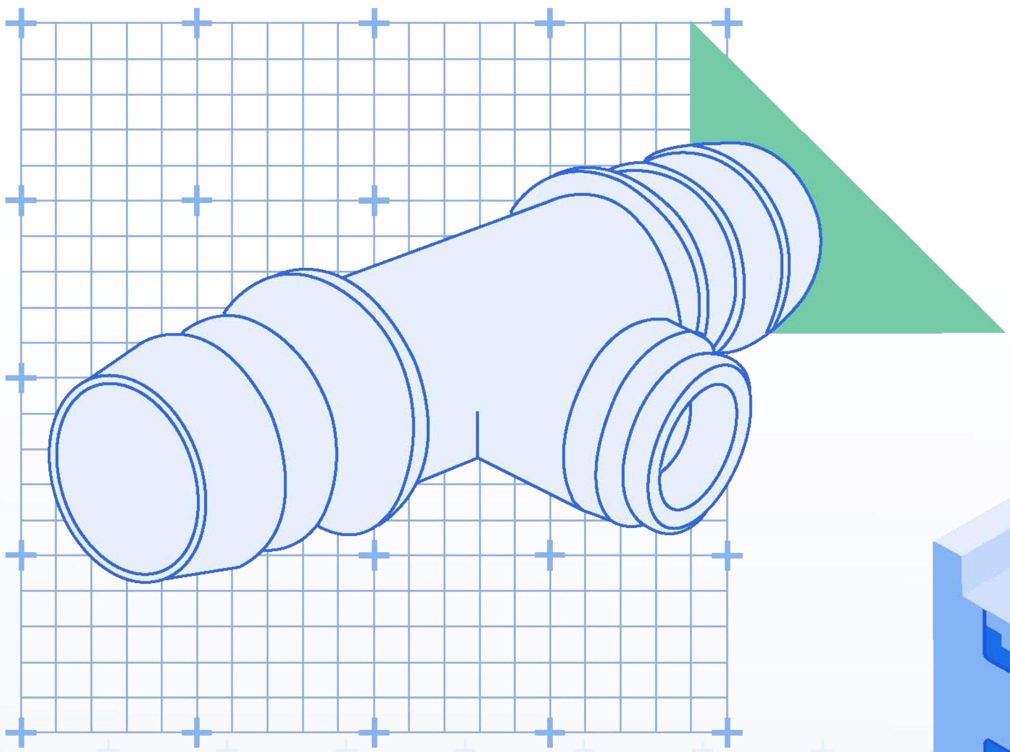
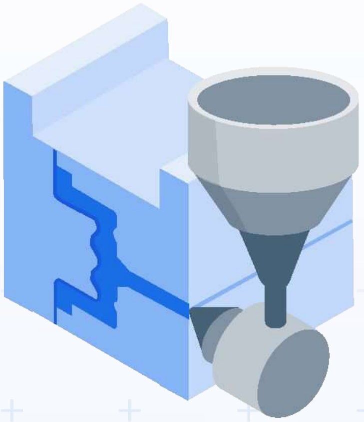
Design Guidelines and Best Practices for Injection Molding





Authored by the Xometry Team

Table of Contents

Introduction to Injection Molding ... **3**
Overview of Injection Molding ... 4
Types of Injection Molding Molds ... 6
Applications and Benefits of Injection Molding ... 11
Material Selection ... **13**
Overview of Injection Molding Materials ... 14
Surface Finishes & Textures ... **22**
Overview of Finishes in Injection Molding ... 23
SPI Surface Finishes ... 24
VDI Surface Finishes ... 27
Mold-Tech Textured Finishes ... 28
Injection Molding Finishes Guide ... 29
Injection Molding Design Guidelines ... **30**
Draft Angles ... 31
Wall Thickness ... 33
Radii and Fillets ... 35
Bosses, Ribs & Gussets ... 37
Living Hinges ... 39
Snap-Fits ... 43
Texts & Symbols ... 50
Gate Design ... 52
Parting Line ... 54
Dealing With Undercuts ... 56
Plastic Part Tolerances ... 61
Common Defects & How to Prevent Them ... **63**
Warping ... 64
Sink Marks ... 65
Knit Lines ... 66
Drag Marks ... 67
Splay ... 68
Injection Molding in Action ... **69**
Injection Molding With Xometry ... **77**
Additional Resources ... **85**

DESIGN GUIDE

Introduction to Injection Molding

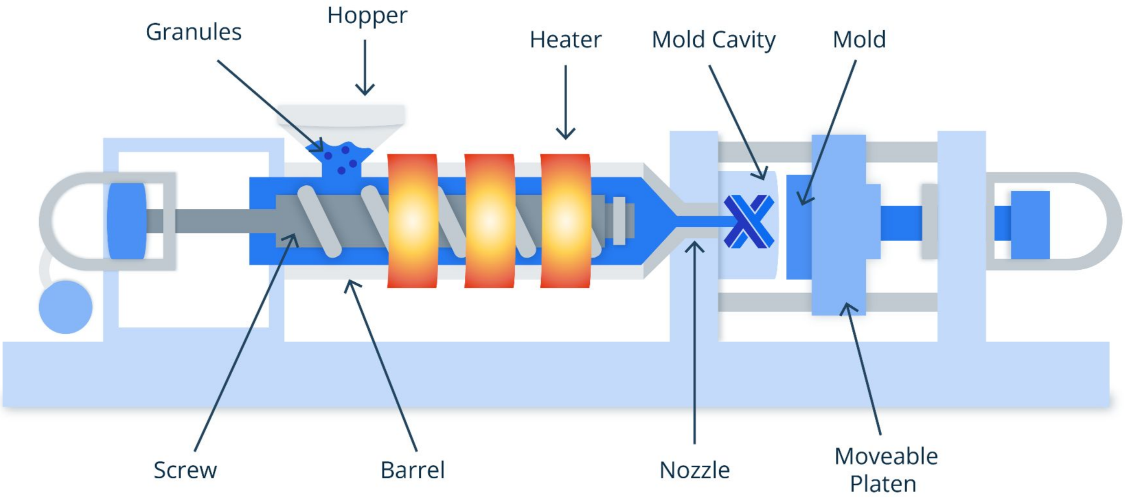
Overview of Injection Molding

Injection molding is a versatile and powerful production process that can quickly and efficiently manufacture parts in large quantities. As the name suggests, molten resin is injected into molds under high pressure and allowed to cool under controlled conditions to form the intended part or product. Once the resin cools and solidifies, maintaining its shape, the core and associated parts retract from the cavity, and ejector pins push the manufactured part out of the mold, and the process repeats.

It can be used to create innovative and original designs, achieve advanced performance and tight tolerances, and improve cost-effectiveness at the same time. Injection molding has become one of the most popular manufacturing technologies in this century, where we see mass manufacturing projects almost every day.

Today, it plays a vital role in the manufacturing of millions of products including sunglasses, television bezels, insulation seals, smartphones, electromechanical enclosures, and more.

Injection Molding Machine



To ensure the most success when working with injection molding, you should make sure to follow Design for Manufacturing (DFM) best practices. DFM refers to special engineering practices that make your product ideal in terms of its look and functionality while eliminating common defects encountered in the manufacturing process. By incorporating DFM practices, carefully planning product and tool design, and selecting the right material, finish, and other details, you will be on a path to a perfect product.

We want your tooling and molded components to be manufactured in a way that leaves as little room for error as possible, which is why we've constructed this guide to help you. We'll discuss topics related to manufacturing with plastic injection molding such as:

- ➔ Understanding the types of molds
- ➔ Choosing the right material
- ➔ Design criteria for the injection molds and molded components
- ➔ Practical design tips to reduce manufacturing time and costs
- ➔ Common defects and how to avoid them

Types of Injection Molding Molds

When considering the costs of manufacturing with injection molding, the cost of manufacturing the mold is the most important. High unit prices can be avoided with a meticulous planning process and the right decisions in mold manufacturing.

Molds are typically manufactured using CNC machining by utilizing steel or aluminum material. Long-lasting mass-manufacturing molds with complex designs are manufactured from stainless steel using methods such as wire and die sinking EDM. Various materials can be tested, and hybrid manufacturing technologies or post-processes can offer solutions depending on the requirements of the project.

Choosing the right type of mold is critical as it affects the service life and quality of your product, while also driving manufacturing volume and time.

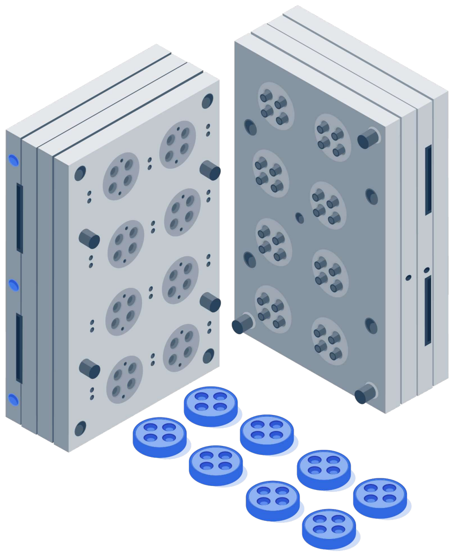
Single-Cavity Mold

Single cavity molds are designed to produce one part per molding cycle. They consist of a single cavity shaped like the desired part, which is filled with molten plastic during the injection process. Single cavity molds are often used for low to medium production volumes, prototypes, or when manufacturing complex or high-precision parts.

While production rates with them are slower compared to multi-cavity molds, they are typically less expensive to design and manufacture, making them ideal for specialized or smaller-scale applications.

Types of Injection Molding Molds

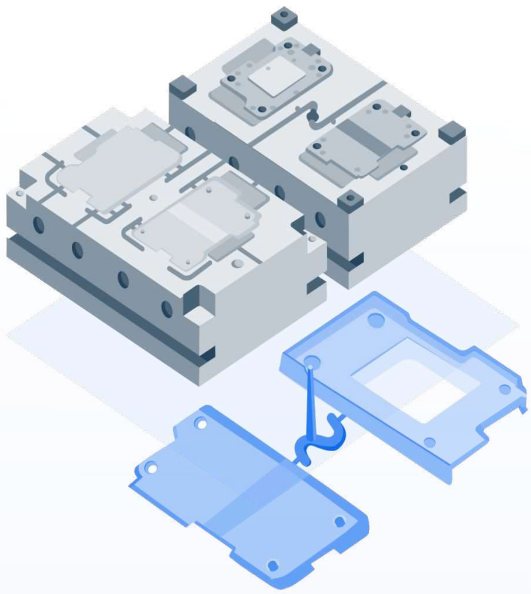
Multi-Cavity Mold



Multi-cavity injection molds have the ability to simultaneously manufacture multiple identical parts in the same cycle. It contains multiple cavities, each shaped like the desired part, allowing for increased production output and efficiency. Multi-cavity molds are commonly used for high-volume manufacturing where consistency and speed are critical.

Although they are more expensive and complex to design and fabricate than single cavity molds, they significantly reduce production costs per part by optimizing cycle times and output.

Family Mold



Different parts with similar volumes will have similar filling profiles and shrinkage ratios. You can take advantage of this by manufacturing these parts in a family mold. This saves you the cost of manufacturing separate molds for each part. Plastics of the same material and color can be manufactured with family molds, which are cost-effective for low-volume orders.

Family molds can also be modified to run one cavity as a time if necessary; in this case the benefit is cost savings by consolidating cavities all in one tool in exchange for lower throughput since parts cannot be manufactured in parallel.

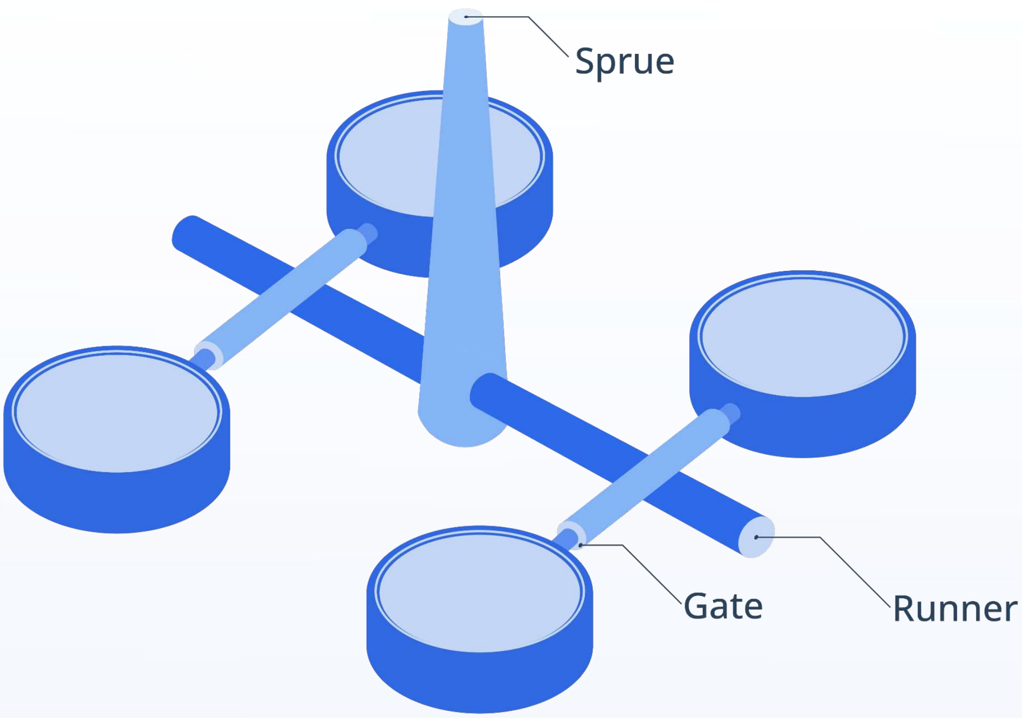
Family Mold

Keep in mind that family molds have a higher defect rate due to the complexity of the mold and the presence of multiple mold models in a single manufacturing cycle.

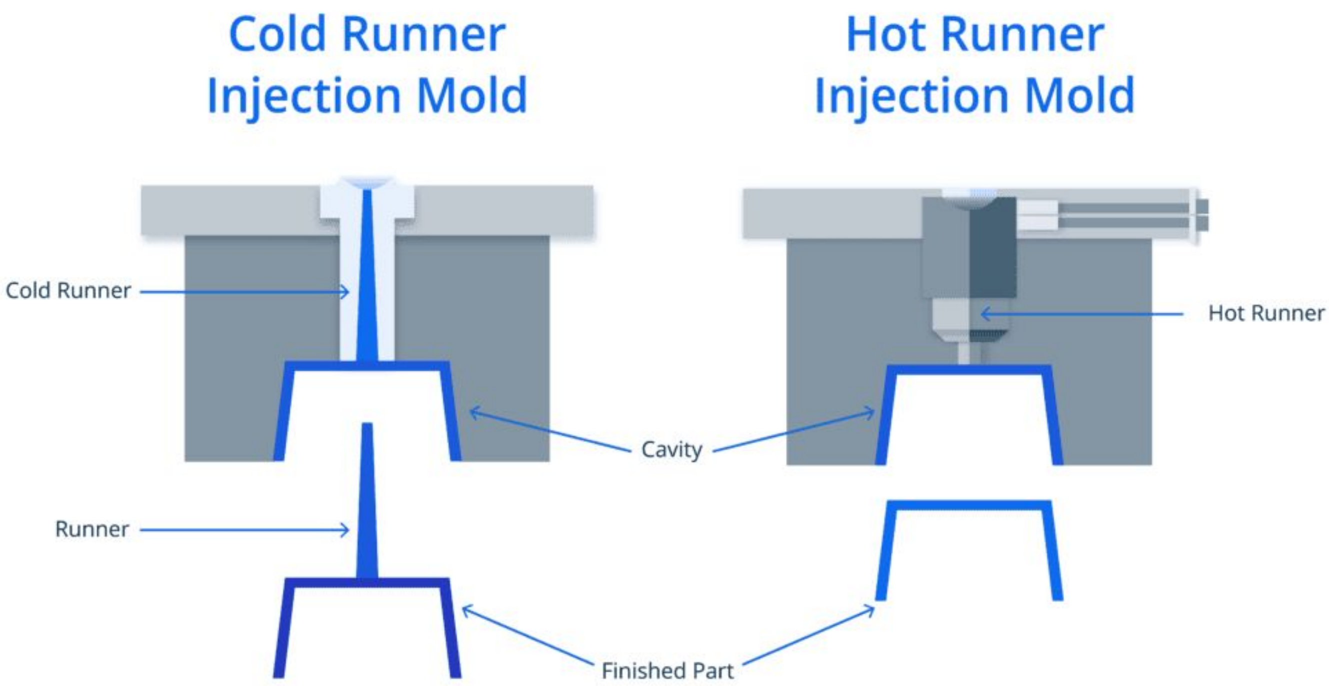
Runners

Runners are the channels in a mold that lead molten plastic from the nozzle to the part cavity. It can sometimes be a single stem, known as a sprue, or could be a branching arm from the sprue to the part. It's important to understand the different types of runner systems used in injection molding, as they have a direct impact on part quality, production efficiency, and cost.

Runner systems are split into two categories: hot runner and cold runner systems, each with its strengths and weaknesses to consider.



Types of Runner Systems



Cold Runner Systems

In cold runner systems, the molten plastic cools and solidifies together with the part in the mold cavity. Cold runner systems are ideal for low-volume manufacturing and are compatible with any plastic material. However, surface marks called *gate vestiges* are more likely to appear on the parts as the solidified and connected runners need to be removed through post-processing.

- ➔ Suitable for low-volume manufacturing
- ➔ Low mold costs
- ➔ Trimming can be manual or automated
- ➔ Compatible with every material
- ➔ Surface marks from removed runners more likely to appear

Types of Runner Systems

Hot Runner System

In hot runner systems, the molten plastic is kept at an optimal temperature until it is transferred to the mold cavity. Cooling or solidification of the runner does not occur during the manufacturing cycle. Hot runner systems can be heated externally or internally. Externally heated systems should be used if heat-sensitive resins are used. There is a significant reduction to waste and post-processing associated with this system; however, it is more expensive and complex to set up.

- ➔ Suitable for high-volume manufacturing
- ➔ Low amount of waste
- ➔ Eliminates the need for trimming
- ➔ Finer control over pressure and temperature
- ➔ Higher mold and maintenance costs

Applications and Benefits of Injection Molding

We briefly mentioned some injection molding benefits and applications. Still, let's dive deeper to understand why it's become one of the most utilized processes for plastic part production in the world of manufacturing.

Cost Efficiency

Perhaps one of the biggest factors driving the popularity of injection molding is its ability to reduce part costs, particularly at high volumes. Although tooling and molds do not come cheap, they can be made to last for tens to hundreds of thousands or even a million or more cycles. Much of the molding process is highly automated, leading to large production volumes, lower labor costs, and minimal waste. Depending on part size, geometry, and material, it can take only a matter of seconds for the process to churn out a completed piece.

High-Volume Production Suitability

The ability to use tooling that can last for thousands or even millions of cycles, combined with automation and rapid cycle times sets injection molding up to be a production powerhouse. On top of that, the process is highly repeatable ensuring a consistent product. It's not just limited to higher quantities though, injection molding can be equally effective at lower volumes as well.

Xometry offers quick-turn and prototype molding services, in addition to our high-volume offerings, enabling you to scale production throughout your product lifecycle.

Applications and Benefits of Injection Molding

Material Versatility

Another great benefit of injection molding is its versatility in materials. Engineers and manufacturers alike can choose from a diverse array of materials to suit the specific needs of the product. From flexible and impact-resistant materials such as polypropylene to rigid and heat-resistant options like polycarbonate, the process can accommodate different properties such as strength, durability, chemical resistance, and transparency.

In addition to this, additives, such as colorants, UV stabilizers, reinforcing fibers, and more, can be mixed in to further customize or enhance performance.

In the next section, we'll explore materials in more detail, as this is often one of the first and most important considerations to make when designing for injection molding.

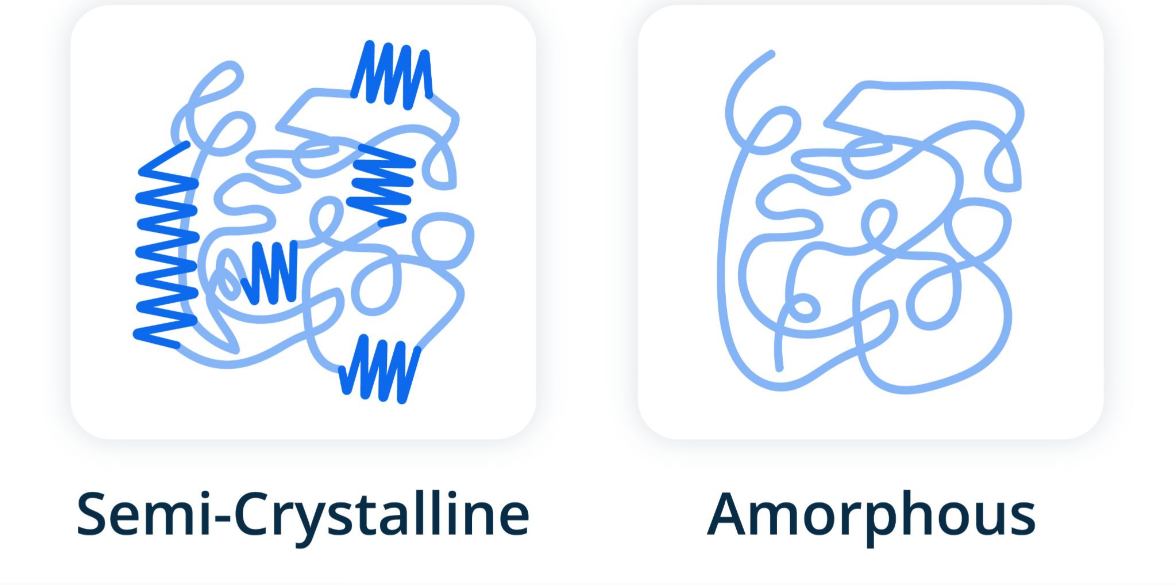
DESIGN GUIDE

Material Selection

Overview of Injection Molding Materials

Plastic injection molding has a wide variety of materials. The chosen material affects the product's strength, flexibility, heat resistance, durability, and cost, as well as how it flows and cools during the molding process. Different plastics have unique properties that impact mold design, cycle times, and even secondary processes like assembly or finishing. This is why we recommend choosing a material early on during the process.

We can generally categorize plastic materials into two groups: *semi-crystalline polymers* and *amorphous polymers*. The key difference between them is in their molecular structures.



Semi-crystalline polymers have highly ordered molecular regions mixed with some amorphous areas, giving them higher chemical resistance, wear resistance, and overall better mechanical properties; however, they can be more challenging to process. These include the polyethylene family (LDPE, HDPE, UHMW-PE), Polypropylene, nylon, acetal, and fluoropolymers.

Overview of Injection Molding Materials

Amorphous polymers have a more random, disordered molecular arrangement. This makes them easy to process and better for precise molding applications. The random nature of their molecular structure also helps prevent light scattering, allowing for transparent and translucent parts. Although they can fit a wide range of applications, they are generally less chemical and wear-resistant. Common amorphous polymers include polycarbonate, acrylic (PMMA), PETG, ABS, polystyrene, and polysulfone.

The table below summarizes the key characteristics of these two main thermoplastic categories.

Semi-Crystalline Thermoplastics	Amorphous Thermoplastics
Highly ordered molecular structure	Irregular molecular structure
Opaque	Transparent, Translucent, or Opaque
Higher dimensional instability	Greater dimensional stability
High heat and wear resistance	Generally less resistant to heat and wear
Distinct and higher melting points	Gradually soften; ideal for thermal shaping
High chemical resistance	Less resistant to chemicals, but suitable for bonding with adhesives
Lower coefficient of friction	Greater coefficient of friction

Common Injection Molding Materials

We understand there are many resins to choose from, and it can be daunting to pick one, especially if you are less familiar with the options! For your convenience, we've put together the table below that compares some of the most common injection-molded materials, their characteristics, applications, and general cost range.

Commodity Plastics			
Material	**Characteristics**	**Common Applications**	**Relative Cost**
Polypropylene (PP)	Good chemical and moisture resistance; won't degrade when exposed to moisture or water. Good elasticity and high impact strength.	Integral hinges or living hinges, fans, snap-over lids, medical pipette tubing.	€
Polystyrene (PS)	Lightweight, relatively inexpensive, and resistant to moisture and bacterial growth Non-toxic, odorless Clear, hard, and brittle.	Plastic utensils, containers, optics, toys.	€
Acrylonitrile Butadiene Styrene (ABS)	Lightweight, chemical resistant, and environmentally friendly. High impact and tensile strength.	Fuel tanks, connector insulators, and food containers.	€
Acrylic (PMMA)	Strong, lightweight, shatter-resistant, optically clear, and UV and weather-resistant.	Light pipes, lenses, light shades, optical fibers, signs.	€
Polyethylene (PE)	Chemically resistant, food-safe, high impact resistance, excellent low-temperature resistance, excellent electrical insulation.	Electrical insulation, household containers, toys, buckets, and bottles.	€€
Polyvinyl Chloride (PVC)	Weather, chemical, corrosion, and shock resistant. Available in both rigid (shore D) and rubber-like (shore A) options.	Pipes, window frames, bottles, electrical distribution boxes.	€€

Common Injection Molding Materials

Engineering Plastics			
Material	**Characteristics**	**Common Applications**	**Relative Cost**
Polycarbonate (PC)	Strong, lightweight, and naturally transparent. Remains stable over a wide temperature range.	IAutomotive, electronics, telecommunications.	€
Polycarbonate-Acrylonitrile Butadiene Styrene (PC-ABS)	High impact, bending, and heat resistance. Slightly more flexible than standard polycarbonate.	Dashboards, electronic housings, medical device casings, and power tool bodies.	€
PA (Aliphatic Polyamides) / Nylon	Excellent heat resistance, high abrasion resistance, high tensile strength, and good fatigue resistance.	Medical and chemical instrumentation, tableware and catering, HVAC and fluid handling, electrical and lighting.	€€
Polybutylene Terephthalate (PBT)	High tensile strength, stiffness, and excellent stain resistance. Good creep resistance and low moisture absorption.	Slide bearings, gears, and cams, hair dryer nozzles, vacuum cleaners, handles, and knobs for electrical cookers.	€€
Acetal or Polyoxymethylene (POM) / Delrin	High tensile strength, rigidity, and toughness. Good impact strength, even at low temperatures. Low moisture absorption and excellent machinability, creep resistance, and dimensional stability.	Gears, pumps and pump impellers, conveyor links, soap dispensers, fan and blower blades, automotive switches, electrical switch components, buttons, and knobs.	€€

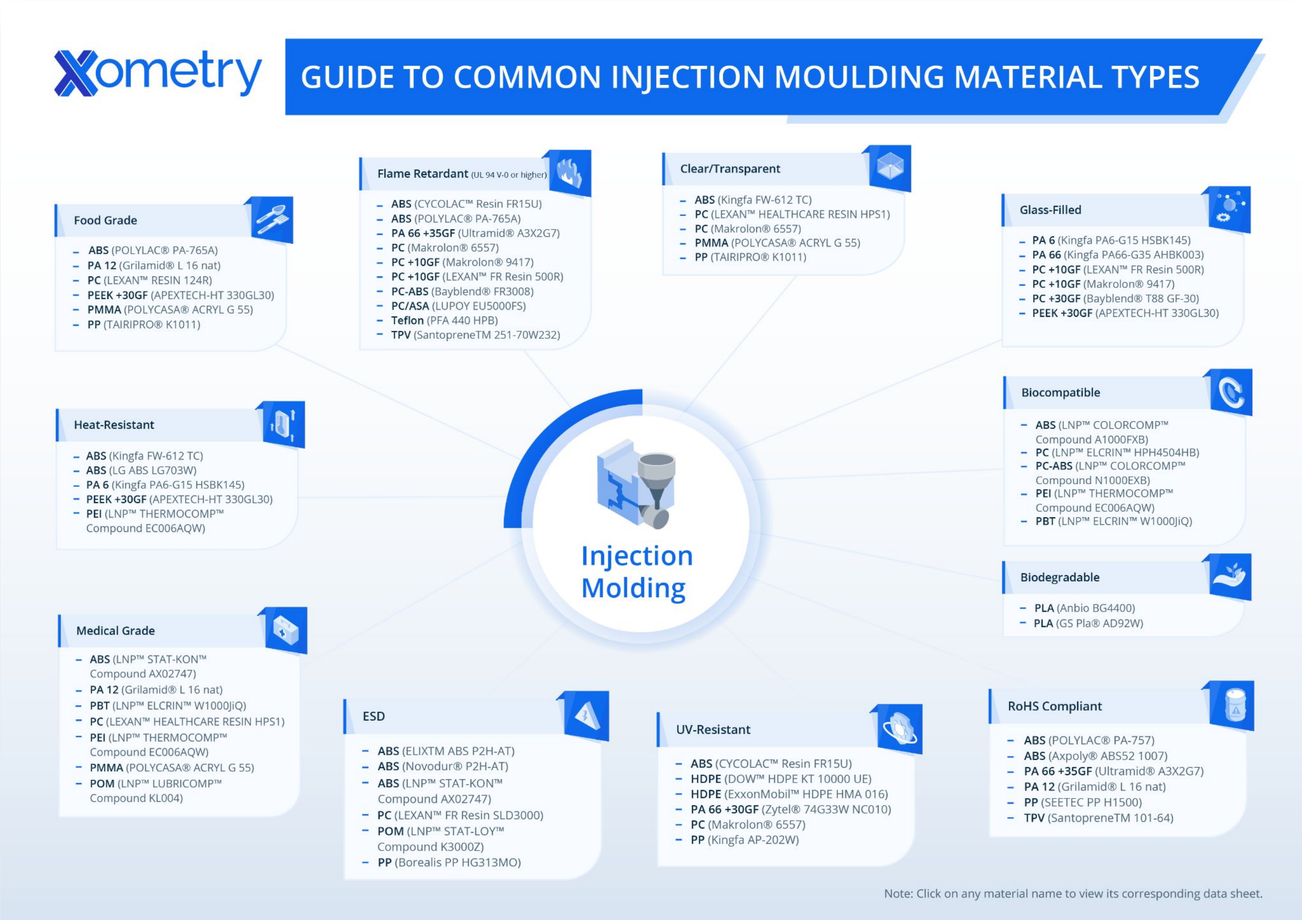
Common Injection Molding Materials

Elastomers & Rubber-Likes			
Material	**Characteristics**	**Common Applications**	**Relative Cost**
Thermoplastic Elastomer (TPE)	High impact strength, flexible, and rubber-like material. Excellent resistance to chemicals & weathering.	Medical equipment, power tools, housewares, mobile phones, computer mice, belts.	€
Thermoplastic Polyurethane (TPU)	Rubber-like elasticity with good load-bearing capabilities. Excellent abrasion resistance, good adhesion to substrates, is flame retardant, has very good microbial resistance. Exhibits properties like durability, flexibility, and excellent tensile strength.	Wire and cable jacketing, hose and tube, in adhesive and textile coating applications.	€
Thermoplastic Vulcanizate (TPV) / Santoprene	Synthetic rubber-like elastomer with high versatility due to temperature resistance, compression, and elasticity.	Tires, hoses, grips, handles, medical tubing, gaskets and seals.	€€

High-Performance Plastics			
Material	**Characteristics**	**Common Applications**	**Relative Cost**
Polyetherimide (PEI) / ULTEM	Extremely high heat and flame resistance and low smoke production. High dielectric strength and good stiffness and hardness.	Medical and chemical instrumentation, tableware and catering, HVAC and fluid handling, electrical and lighting.	€€€
Polyether Ether Ketone (PEEK)	High tensile strength and high resistance to chemicals and thermal degradation. Total biocompatibility, UV resistance, pure radiolucency, gamma-ray resistance, and low/no toxicity.	Medical implants, bearings, piston parts and pumps, electric insulation, valves and seals.	€€€

Material Selection Infographic

To make material selection even easier for you, we've put together this infographic to highlight the most common materials and the key properties they excel in.



While Xometry offers a wide array of resins and can even work with customer-supplied materials, we recommend sticking to the options listed thus far to ensure the fastest quote response and material availability. If you're interested in supplying your own resins or want to use one not on this list, just let us know and we'll be happy to work that out with you.

Material Additives and Fillers

Fillers and additives play a crucial role in enhancing the mechanical, chemical, and cosmetic properties of injection-molded plastics. Reinforcing materials such as glass and carbon fibers can be added to many resins to improve strength, stiffness, and impact resistance, though they may also make parts slightly more brittle. Other fillers, like talc, increase hardness, while glass beads and fused silica help reduce flexibility, minimize warpage, and control shrinkage.

Additionally, additives such as UV stabilizers, static dissipating agents, and flame retardants can be incorporated into the resin compound to enhance performance based on specific application needs.

Careful selection and testing of fillers and additives are essential to achieving optimal results in mass manufacturing. Thorough analyses and prototype testing with different plastics ensure the right combination and concentration of additives, leading to a more efficient and trouble-free production process. It is important to note that fillers can alter the appearance of a part, sometimes causing visual effects like flow marks as the fibers orient in the direction for the molten resin filling the cavity.

These fiber orientation marks are typically prominent around gate locations, forming a circular shape, and can be mitigated through gate placement and features that facilitate even material flow.

Surface of a part made with filled material

Material Additives and Fillers

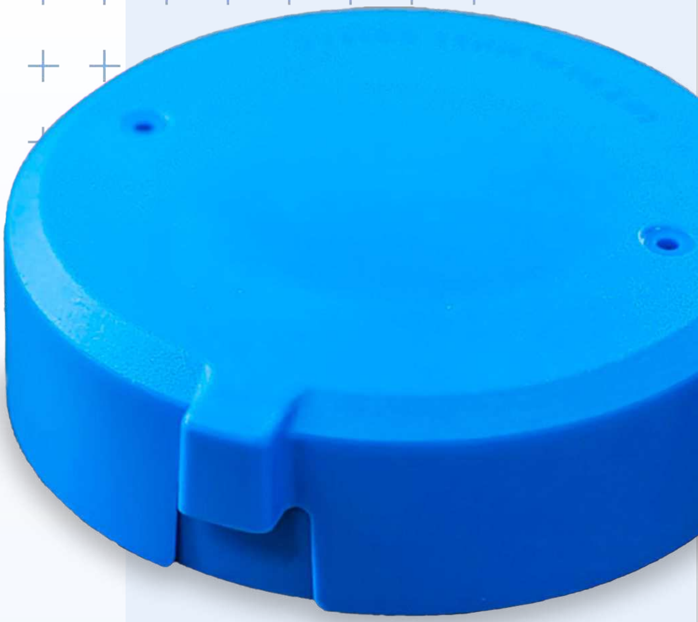
There are several types of fillers and additives, each serving distinct purposes:

- **Glass and carbon fibers** enhance mechanical performance by improving tensile strength, impact resistance, and crack resistance. They are commonly used in plastics such as ABS, PEEK, PET, Nylon, and PC.
- **Ceramic fillers** improve heat conductivity, making them suitable for high-temperature applications.
- **Calcium carbonate and talc** are cost-effective fillers that help reduce raw material expenses without compromising mechanical performance.
- **Colorants** are added to achieve specific colors and aesthetic effects in molded parts.
- **Thermal-resistant additives**, such as phosphorus-based flame retardants and UV stabilizers, increase durability and protect against heat and environmental exposure.

The right combination of fillers and additives can help product designers and manufacturers optimize and enhance existing material properties and meet specific design and functional requirements.

DESIGN GUIDE

Surface Finishes & Textures



Overview of Finishes in Injection Molding

Injection molding produces highly precise parts that replicate even the smallest mold imperfections. Molds, typically made from aluminum or steel, can transfer machining marks to molded parts unless removed through bead blasting or polishing. While interior, non-visible surfaces can often retain these marks without issue, visible areas usually require additional finishing, which increases tooling cost and lead time.

Surface finishes serve both functional and aesthetic purposes, ranging from glossy and matte to rough or textured. These options can hide defects like sink marks and warping, enhance grip, or improve part strength. Deciding early which surfaces require finishing and to what extent can help reduce unnecessary or excessive finishing steps, minimizing overall production costs.

While injection-molded parts typically don't require post-processing, they can be customized with applications like ultrasonic welding, laser engraving, or painting. The ideal surface finish should balance aesthetics, functionality, tolerance, and cost for the best final product.

SPI Surface Finishes

The most common finishes used include those from the Society of the Plastics Industry (SPI), which are a set of standard mold finishes starting from heavily polished to coarse matte finishes.

Where polished textures can increase a parts' cosmetic reflection or transparency on transparent polymers, matte finishes can help provide more subdued tones and even help prevent fingerprints on handled products (e.g., your laptop's frame and keyboard).

The various surface finish grades can be achieved through machine texturing or manual polishing. The SPI groups different surface finishes into four grades, as outlined below.

High Gloss Finishes (Grades A1, A2, and A3)

These grades exhibit the highest level of mold polishing with a minimum roughness of 0.012 micrometers. The molded parts will have a highly glossy or transparent finish on transparent plastics. The high gloss is achieved through post-processes such as diamond buff polishing.

However, not all materials can have a grade-A finish because some plastics have high resistance to abrasion. For example, you will never find a high-gloss finish on parts molded with TPU.

SPI Surface Finishes

Semi-Glossy Finishes (Grades B1, B2, and B3)

Surfaces with semi-gloss finishes have a minimum roughness of 0.05 micrometers. The grade-B finishes are achieved using grit sandpaper and are the preferred injection molding surface finish for consumer products. Molded parts with this level of finish are appealing to the eye, with no noticeable surface imperfections.

Matte Finishes (Grades C1, C2, and C3)

Matte finishes have a minimum surface roughness of 0.35 micrometers and are only suitable for parts that are not expected to be highly aesthetic but still require relatively smooth surfaces. This SPI grade still offers a smooth surface with no visible tooling marks or other surface imperfections.

Textured Finishes (Grades D1, D2, and D3)

Some parts are intentionally molded with rough textures to improve functionality or performance. Rougher surfaces are typically achieved through sandblasting, which creates a textured finish classified as SPI grade D. This added texture can provide benefits, such as enhanced grip or increased friction. Other benefits of texturing the mold include:

- Hides surface imperfections
- Strengthens injection molds
- Enhances slip resistance
- Improves adhesion property of the molded parts

SPI Finish Grades Comparison Table

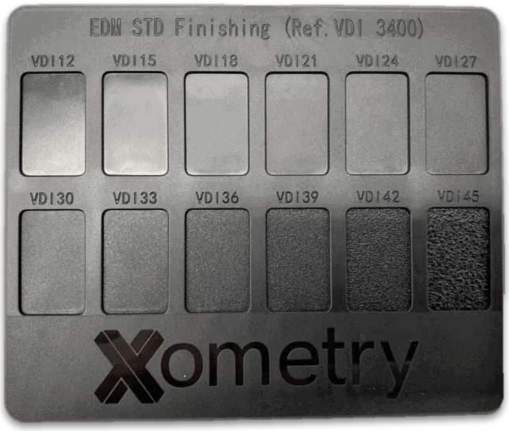
The table below can be used to compare the various SPI surface finish grades by their applications, surface roughness, and compatible material options.

SPI finish	Description	Applications	Surface roughness (Ra µm)	Suitable material examples	Added Cost
A-1	Grade #3, 6000 Grit Diamond Buff	High polish parts	0.012- 0.025	Acrylic	€€€€€
A-2	Grade #6, 3000 Grit Diamond Buff	High polish parts	0.025-0.05	Acrylic, PC	€€€€
A-3	Grade #15, 1200 Grit Diamond Buff	High low polish parts	0.05-0.10	ABS, Acrylic, PS, Nylon, PC	€€€€
B-1	600 Grit Paper	Medium polish parts	0.05-0.10	ABS, Acrylic, PP, PS, HDPE, Nylon, PC	€€
B-2	400 Grit Paper	Medium polish parts	0.10- 0.15	ABS, Acrylic, PP, PS, HDPE, Nylon, PC	€€
B-3	320 Grit Paper	Medium low polish parts	0.28-0.32	ABS, Acrylic, PP, PS, HDPE, Nylon	€€
C-1	600 Stone	Low polish parts	0.35-0.40	ABS, Acrylic, PP, PS, HDPE, Nylon, TPU	€
C-2	400 Stone	Low polish parts	0.45-0.55	ABS, Acrylic, PP, PS, HDPE, Nylon, TPU	€
C-3	320 Stone	Low polish parts	0.63-0.70	ABS, Acrylic, PP, PS, HDPE, Nylon, TPU	€
D-1	Dry Blast Glass Bead	Satin finish	0.80-1.00	ABS, PP, PS, HDPE, Nylon, PC, TPU	€
D-2	Dry Blast #240 Oxide	Dull finish	1.00-2.80	ABS, PP. PS, HDPE, Nylon, TPU	€
D-3	Dry Blast #24 Oxide	Dull finish	3.20-18	ABS, PP, PS, HDPE, Nylon, TPU	€

VDI Surface Finishes

VDI 3400 Surface Finish (commonly known as VDI surface finish) refers to the mold texture standard set by Verein Deutscher Ingenieure (VDI), the Society of German Engineers.

The VDI 3400 surface finish is mainly processed by Electrical Discharge Machining (EDM) when mold machining. It could also be done by the traditional texturing method (like in SPI). Although the standards are set by the Society of German Engineers, they are commonly used among tool makers all over, including North America, Europe, and Asia.



The table below provides a comparison of the various VDI surface finish options.

VDI Value	Description	Appearance	Surface roughness	Added Cost
12	600 Stone	Low polish	0.40 Ra µm	€
15	400 Stone	Low polish	0.56 Ra µm	€
18	Dry Blast Glass Bead	Soft satin	0.80 Ra µm	€
21	Dry Blast # 240 Oxide	Light satin	1.12 Ra µm	€
24	Dry Blast # 240 Oxide	Satin	1.60 Ra µm	€
27	Dry Blast # 240 Oxide	Light matte	2.24 Ra µm	€
30	Dry Blast # 24 Oxide	Medium matte	3.15 Ra µm	€
33	Dry Blast # 24 Oxide	Textured matte	4.50 Ra µm	€
36	Dry Blast # 24 Oxide	Fine grain	6.30 Ra µm	€
39	Dry Blast # 24 Oxide	Medium grain	9.00 Ra µm	€
42	Dry Blast # 24 Oxide	Coarse grain	12.50 Ra µm	€
45	Dry Blast #24 Oxide	Very coarse grain	18.00 Ra µm	€

Mold-Tech Textured Finishes

Standex Engraving Mold-Tech is a standardized mold texturing option typically used for commercial or saleable goods. Mold-Tech is serialized in four series, A through D. Most commonly, the Mold-Tech Series A finishes are used on products as it has a range of fine to coarse matte finishes that do not require laser, masked chemical etching, or other engineered texturing processes.

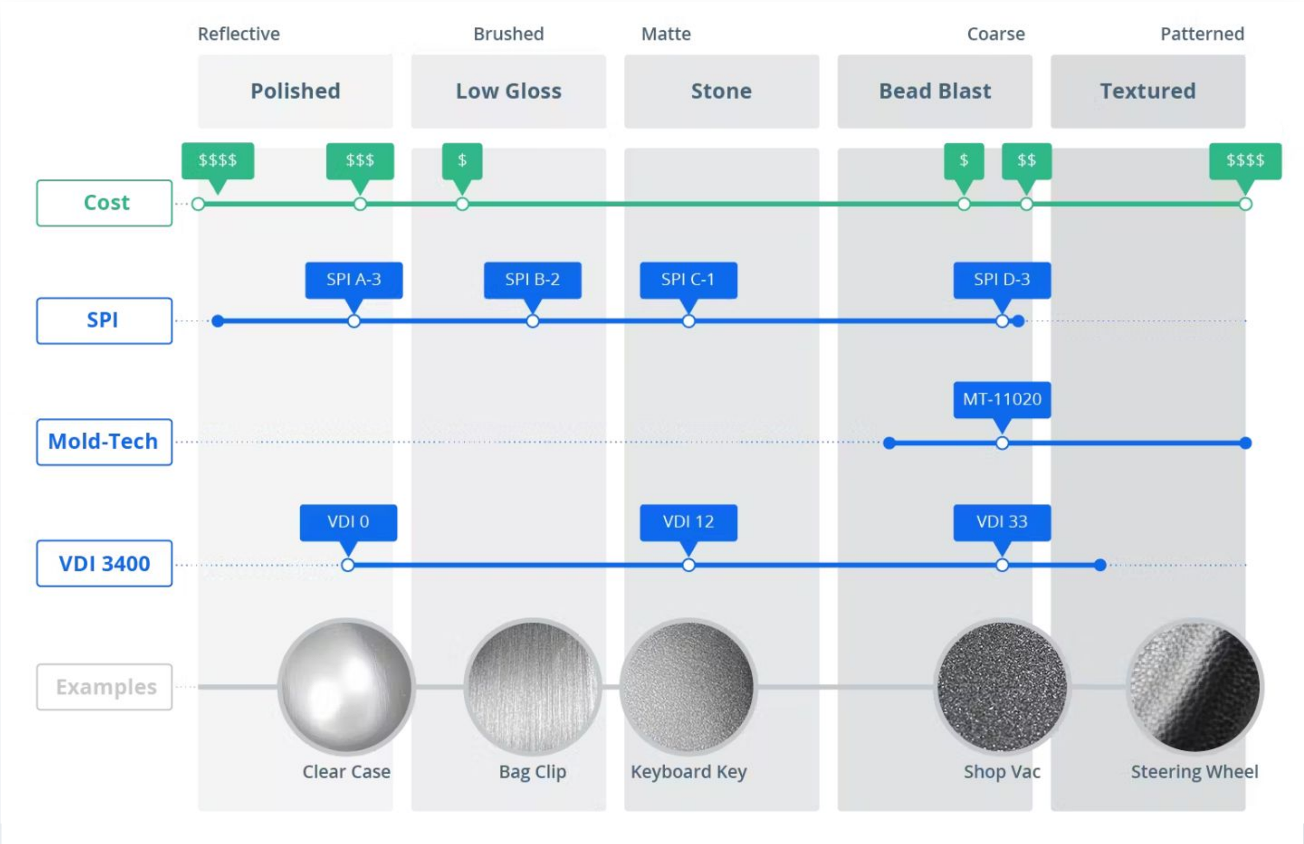
Mold-Tech finishes are categorized by their serial number and texture depth. A more aggressive draft angle is needed with Mold-Tech because mold texturing has more coarse features than SPI finishes, it is recommended to add 1.5° of draft for every .001" of texture depth.

Some examples of common Mold-Tech finishes can be seen below, along with their recommended depth and draft angle.

Mold-Tech Serial Number	Texture Depth (inch)	Appearance	Minimum Draft	Added Cost
MT-11010	.001"	Dull matte finish	1.5°	€€€
MT-11020	.0015"	Coarse matte finish	2.25°	€€€
MT-11030	.002"	Coarse matte finish	3°	€€€

Injection Molding Finishes Guide

Our infographic guide below can help you compare different injection-molded finishes and grades and decide which is best for your project.



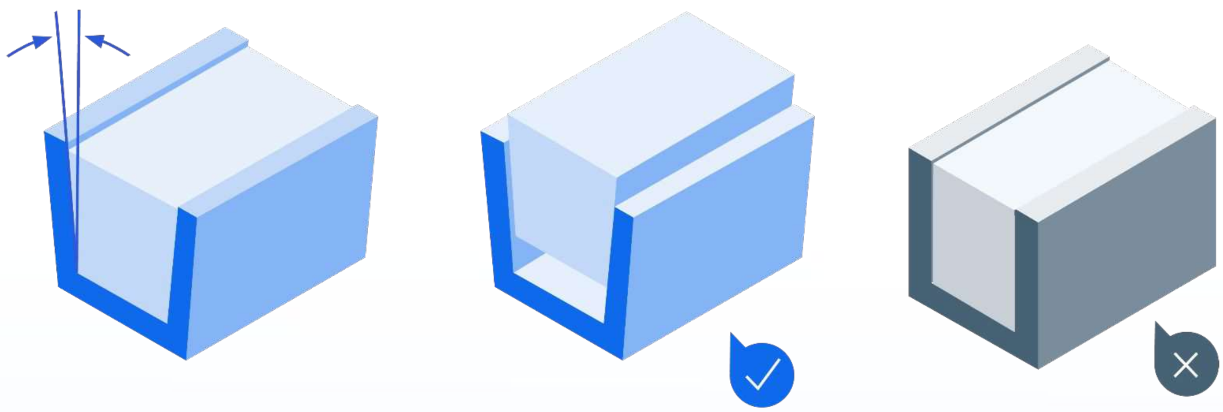
DESIGN GUIDE

Injection Molding Design Guidelines

Draft Angles

When designing injection molded parts, it's important to remember that almost every component needs a draft angle—a slight taper on the walls—to make sure it can be easily removed from the mold. Unlike CNC-machined parts that can have perfectly vertical walls, injection molded parts almost always include a bit of slant.

This extra angle prevents the part from getting stuck in the mold as it cools and contracts. Without the draft, you'd risk excessive force during ejection, which can lower part quality, lead to scrap parts, and may even result in damage to the tooling.



Most CAD software makes adding draft angles a breeze, but it's best to do this during the final stages of your design. Adding draft too early can complicate the design process, and since it alters the part's overall shape, it's crucial to get it right before sending your design off to the molder. This ensures that the intended design isn't compromised and that your molder knows exactly how to incorporate the draft.

Draft Angles

Keep in mind that not all surfaces need the same amount of draft. For example:

- → For parts with standard surfaces or SPI textures, a draft angle of about 1–2° is generally sufficient.
- → If the surface has more pronounced or heavy textures, you might need a draft angle of 3–5° (or even at least 5° for really heavy textures) to avoid issues like scraping or deformation.

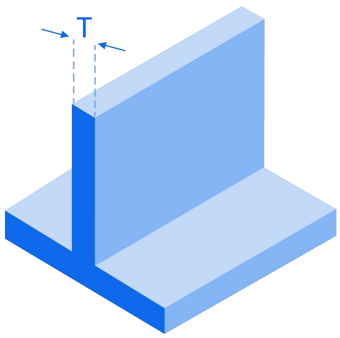
By tailoring the draft angle to the specific features of your part, you help ensure smooth ejection from the mold and maintain the quality of the final product.

Here are some examples of applications and their suggested minimum draft angles:

Application / Surface Description	Minimum Draft Angle
For "near-vertical" requirements	0.5°
General applications	2°
All shutoff surfaces	3°
Lightly textured faces	3°
Medium to heavily textured faces	5°+

Wall Thickness

When designing parts for plastic injection molding, wall thickness is a critical factor affecting mechanical performance, mold life, manufacturing time, aesthetics, and overall cost. The wall thickness must be determined based on the part's performance requirements and the selected resin. For many applications, a wall thickness in the range of **2 mm to 4 mm** is sufficient.



Recommended Wall Thickness by Material

In the chart below you'll find commonly used resins and their recommended wall thickness range:

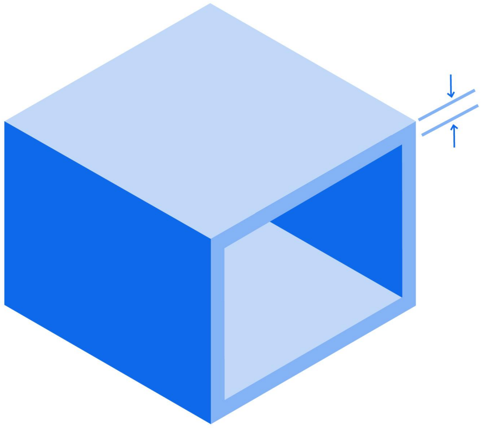
Material	Recommended wall thickness (mm)	Recommended wall thickness (in)
ABS	1.14 mm – 3.56 mm	.045 in - .140 in
Acetal	0.76 mm – 3.05 mm	.030 in - .120 in
Acrylic (PMMA)	0.64 mm – 12.70 mm	.025 in - .500 in
Liquid Crystal Polymer	0.76 mm – 3.05 mm	.030 in - .120 in
Long-Fiber Reinforced Plastics	1.91 mm – 27.94 mm	.075 in - 1.100 in
Nylon	0.76 mm – 2.92 mm	.030 in - .115 in
PC (Polycarbonate)	1.02 mm – 3.81 mm	.040 in - .150 in
Polyester	0.64 mm – 3.18 mm	.025 in - .125 in
Polyethylene (PE)	0.76 mm – 5.08 mm	.030 in - .200 in
Polyphenylene Sulfide (PSU)	0.51 mm – 4.57 mm	.020 in - .180 in
Polypropylene (PP)	0.89 mm – 3.81 mm	.035 in - .150 in
Polystyrene (PS)	0.89 mm – 3.81 mm	.035 in - .150 in
Polyurethane	2.03 mm – 19.05 mm	.080 in - .750 in

Wall Thickness

Importance of Consistent Wall Thickness

Maintaining a consistent wall thickness throughout the part is essential for achieving uniform cooling, which minimizes the risk of warping, sink marks, or other surface defects.

Variations in thickness can lead to aesthetic issues, such as sink marks in thicker areas, so designers often spend significant time “coring out” these sections. Modern CAD tools have streamlined this process by allowing designers to “shell” a part to a specified thickness after the basic shape has been defined.



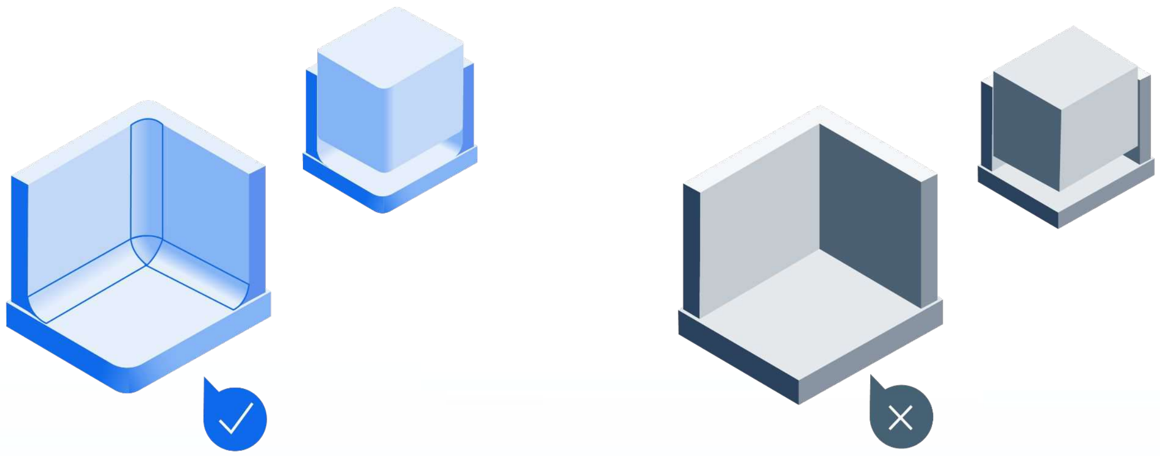
Strengthening Thin Walls

If the design requires thicker walls for added strength or rigidity, consider incorporating design features like ribs and bosses instead. These elements can enhance durability without unnecessarily increasing material usage or cycle time. Excessively thick walls not only waste material but also extend cycle times, making it important to design walls as thin as possible while meeting performance needs.

By carefully balancing wall thickness and using features like ribs, designers can optimize parts for injection molding, ensuring both high quality and cost efficiency.

Radii and Fillets

When designing parts for injection molding, avoiding sharp, steep corners is essential for both manufacturing efficiency and part performance. Sharp external corners often require costly techniques like EDM, whereas using radii, fillets, or chamfers creates smooth transitions that facilitate material flow, reduce stress on the walls, and promote optimal cooling. Rounding features in this way not only prevents the formation of weak points in the molded part but also helps avoid residual stresses that can occur when molten plastic is forced to fill in abrupt corners.

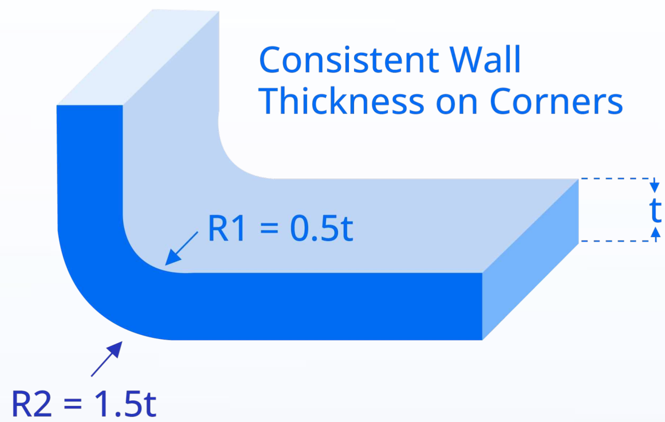


Other considerations for radii and fillets:

- **Design Features**: Vertical details like ribs, bosses, and undercuts should be rounded separately from the part's main corners, while still ensuring that wall thickness remains constant during the rounding process.
- **Transition Guidelines**: For parts with asymmetrical designs or varying wall thickness, smooth out sharp transitions using radii or chamfers. A recommended guideline is to use a transition length of approximately three times the difference in wall thicknesses.

Radii and Fillets

Pro Tip: When rounding corners, ensure that the wall thickness is consistent. This can be achieved by using radii equal to half the wall thickness for internal corners and 1.5 times the wall thickness on outside corners.



Bosses, Ribs & Gussets

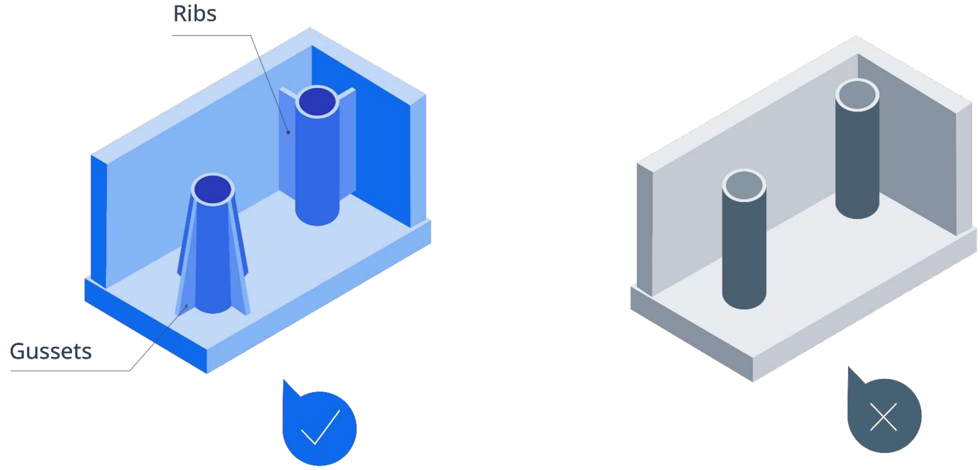
In injection-molded part design, maintaining a minimum wall thickness can simplify the design process and reduce costs, but it also comes with a trade-off in overall strength. To address this, you can incorporate features such as ribs, bosses, and gussets to locally enhance rigidity and durability without unnecessarily thickening walls.

Ribs

Ribs are used to stiffen flat areas or regions that experience external loads. When designing ribs, it is important to consider factors like plastic flow and the potential for sink marks. Our best practices and tips for rib design are as follows:

- ➔ Maintain a rib thickness between 40% and 60% of the main wall thickness.
- ➔ Rib height should be kept below three times the wall thickness.
- ➔ A draft angle of 0.25° to 0.5° is recommended to facilitate mold release.
- ➔ Adding radii of at least 1/4th of the main wall thickness to the rib can help mitigate stress concentrations.
- ➔ Keep the distance between ribs and adjacent walls at least four times the rib thickness to minimize molding defects.

Bosses, Ribs & Gussets



Bosses

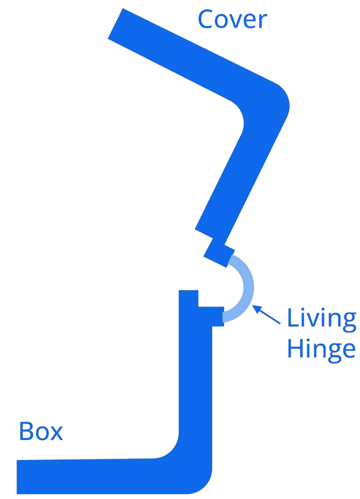
Bosses are cylindrical standoffs molded into the part to accept inserts, self-tapping screws, or pins for assembly or mounting. They function as joint features and can be viewed as circular ribs that contribute to the overall structural strength. For enhanced rigidity, bosses are typically attached to a side wall or a supporting rib rather than being fully integrated into the wall itself. In applications involving self-tapping screws, the outer diameter of the boss is generally designed to be approximately 2½ times the screw diameter. Bosses can also incorporate metal inserts to form durable, long-lasting threads, provided that the specific design criteria are observed.

Gussets

Gussets serve a similar purpose to ribs by connecting a boss to the floor of a part, thereby distributing loads more evenly and providing additional support. Like internal ribs that stiffen flat walls, gussets should not exceed 60% of the overall part thickness; this helps to minimize visible sink marks on the exterior of the part. For instance, in a part with an outer wall thickness of .120 inches, internal ribs or gussets should be designed at approximately .070 inches thick.

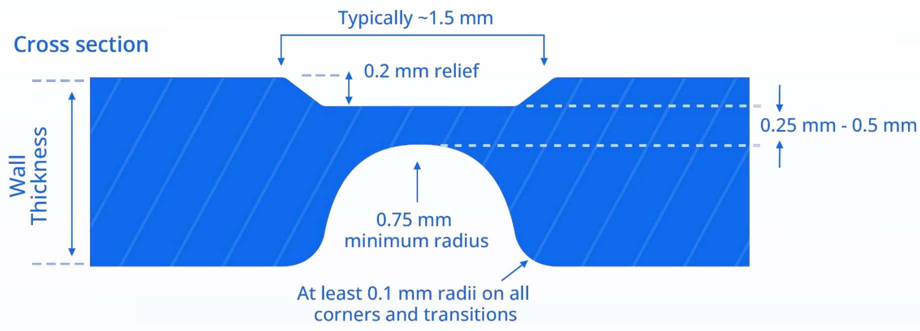
Living Hinges

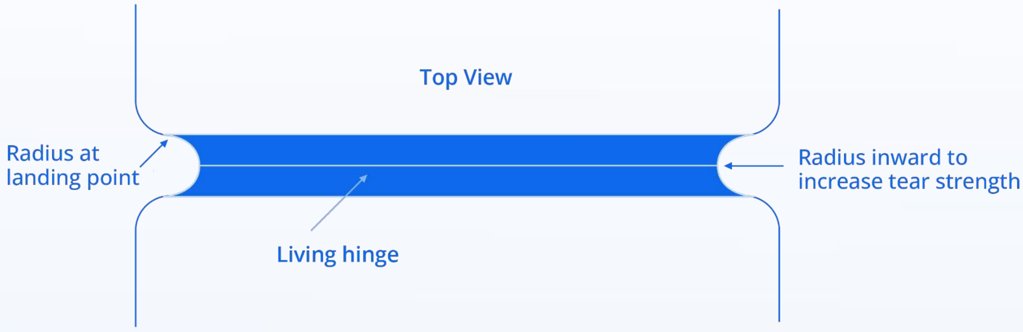
Hinges are a common feature of injection molded parts. A living hinge is a popular type of hinge that consists of a thin plastic section connecting two plastic bodies. This thin section allows the two connected bodies to move 180 degrees or greater. Living hinges are popular as they improve user experience and reduce costs. These satisfying hinges are seen in everyday products such as electronic boxes, packaging lids, and even candy containers.



Other hinge designs require additional components like pins, screws, or metal rods. Hinges made with extra components require additional design considerations such as gates, runners, and other cavities. A living hinge is unique in that it is entirely built into the part. This removes the necessity to purchase other components and design around them.

A properly designed and applied living hinge can function for well over 1 million cycles. Here are some guidelines to get you started off on the right foot with your designs:





Living Hinges

Living Hinge Design Tips

Material: Polypropylene is the most suitable material for living hinges due to its superior flexibility and fatigue resistance, with Polyethylene coming in a close second. Nylon can also be used for living hinges; however, it is less flexible than PP/PE and may fatigue more quickly. Nylon living hinges are advantageous for engineering applications where higher strength and heat resistance are needed.

Thickness: 0.25 mm-0.5 mm is typically suitable for flexible plastics like PP and PE. The ideal thickness will depend on the material chosen, as well as your application, but the goal should be a thickness that is thin enough to flex but thick enough to resist tearing.

Radius: To reduce stress concentrations and improve molding flow, add 0.75mm - 1.50mm radii on either side of the hinge. Sharp corners or abrupt transitions near the hinge could lead to premature failure.

Relief: To promote uniform bending and increased flexibility, a 0.2mm deep relief on the backside of the hinge is recommended. Ensure the relief extends across the width of the hinge. Use radii to smooth the transitions in and out of the relief.

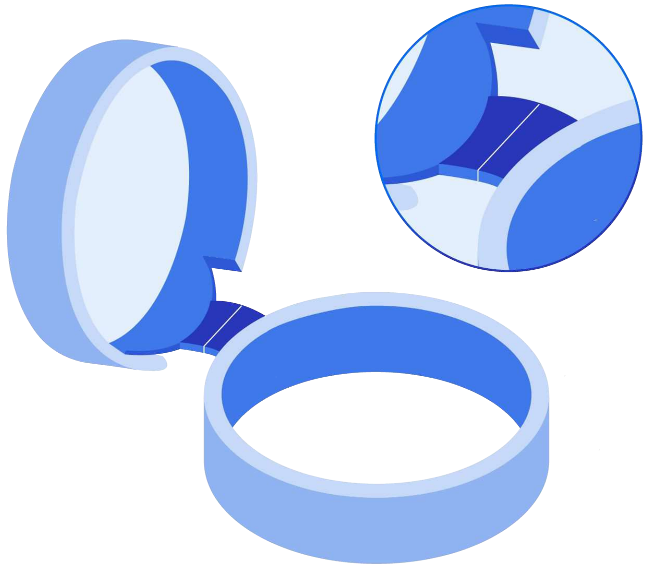
Hinge Length: A hinge length of around 1.5mm is suitable for most designs; however, the length may vary depending on the application. Keep the length practical, as longer hinges do not distribute stresses as evenly, which can lead to decreased durability.ext

Living Hinges

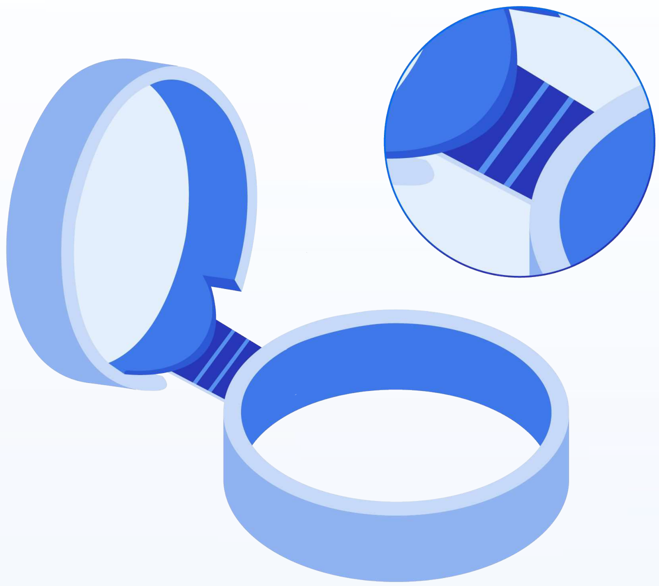
Types of Living Hinges

Not all living hinges are exactly the same. There are four common types of living hinges: flat hinges, double hinges, butterfly hinges, and bi-stable hinges.

Flat Hinge: The most common style of living hinge is a flat hinge. It allows both plastic pieces to lay flat when rotated 180 degrees.

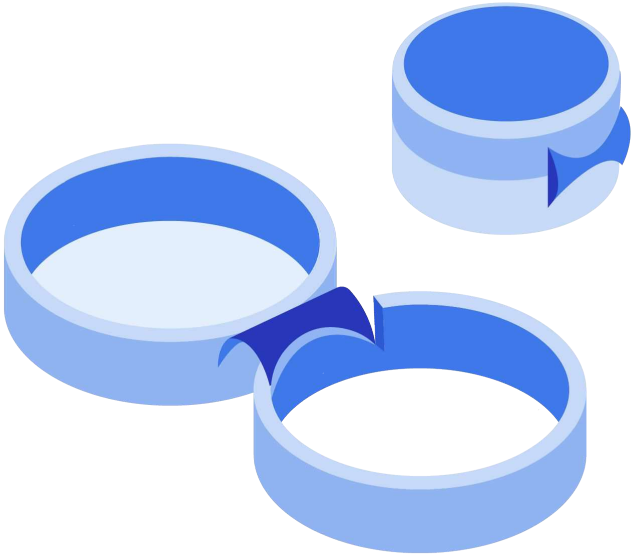


Double Hinge: A double hinge consists of two flat hinges separated by a narrow landing section. These living hinges are useful when you need to create a gap between two folded pieces or you have a design that requires 360 degrees of rotation. CD cases are a common example where a double hinge is implemented.

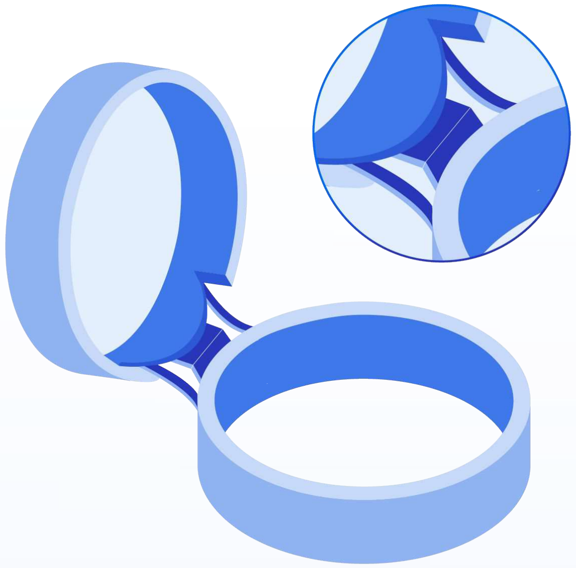


Types of Living Hinges

Butterfly Hinge: A butterfly hinge is most recognizable by its ability to suspend one of its connected pieces in place. The flipping functionality of this hinge allows one piece to span into place while the other piece is anchored in place. This living hinge type is common in ketchup and other condiment containers.



Bi-Stable Hinge: Bi-stable hinges are designed to hold two distinct positions, typically open or closed, without requiring external force to stay in either state. This hinge type typically includes a mechanism that allows it to "snap" into one of two stable configurations.

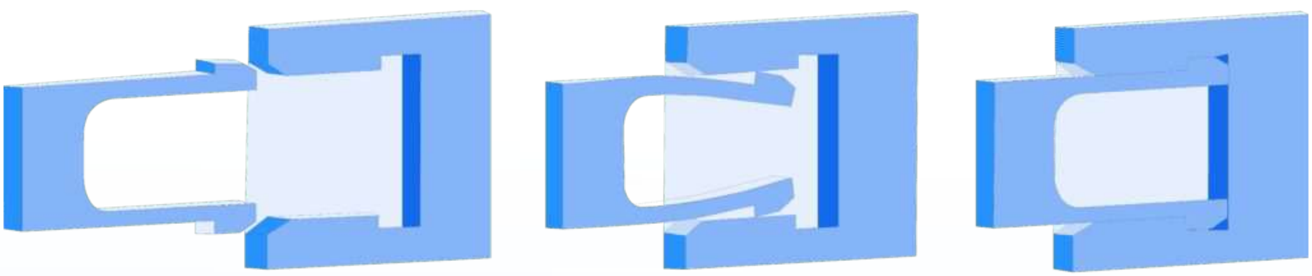


Snap-Fits

A snap fit is a common connection method used for fastening two parts. This connection method typically consists of two pieces: one that has the snap joint and another that has a cavity to receive the joint. Creating snap fits is a simple and economical way to combine two different parts. Snap fits are often designed in one of five different variations, including:

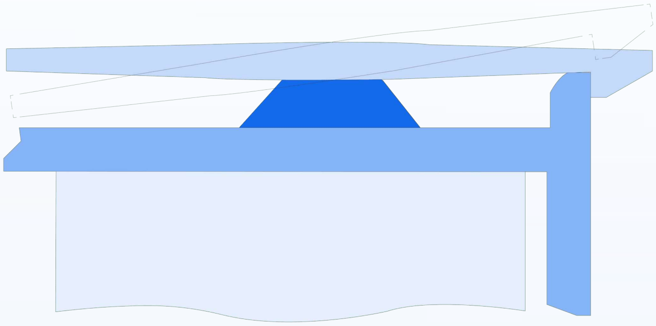
Cantilever Snap-fits

The cantilever snap-fit is the most widely used design of all the snap-fit joints. It consists of a protruding beam (the cantilever) that flexes during assembly to allow the head of the beam to engage in a slot or undercut on the opposing part. Once assembled, the beam returns to its original position, locking the parts together.



Torsional Snap-fits

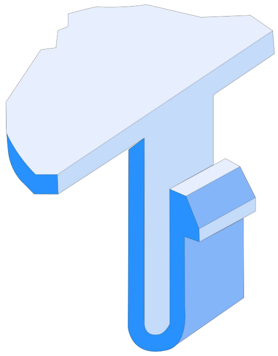
A torsional snap-fit relies on the twisting of a bar rather than bending. The bar acts as a torsional spring, allowing the hook or latch to engage or disengage. This type is ideal when easy separation of the parts is necessary.



Snap-Fits

U-shaped Snap-fits

A U-shaped snap-fit is a double-sided cantilever that provides flexibility from both joint ends. Compared to standard cantilever designs, it offers better alignment and increased flexibility.

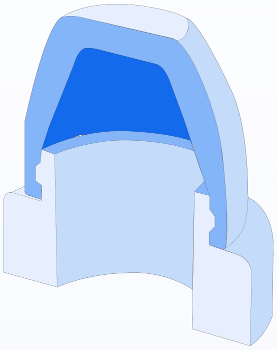


L-shaped Snap-fits

An L-shaped snap-fit provides lateral support, locking parts together through a side-locking mechanism. Unlike cantilever snap-fits, which engage along a vertical axis, L-shaped snap-fits secure parts by applying pressure along a horizontal axis, making them ideal for side-locking applications.

Annular Snap-fits

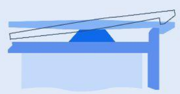
An annular snap-fit forms a circular or ring-shaped joint that locks into place, providing 360° engagement around the part, typically seen in cylindrical components.



Snap-Fits

Quick Guide to Snap-Fit Joints

The table below provides you with a quick comparison of each type of snap-fit to assist you with determining which is best for your design.

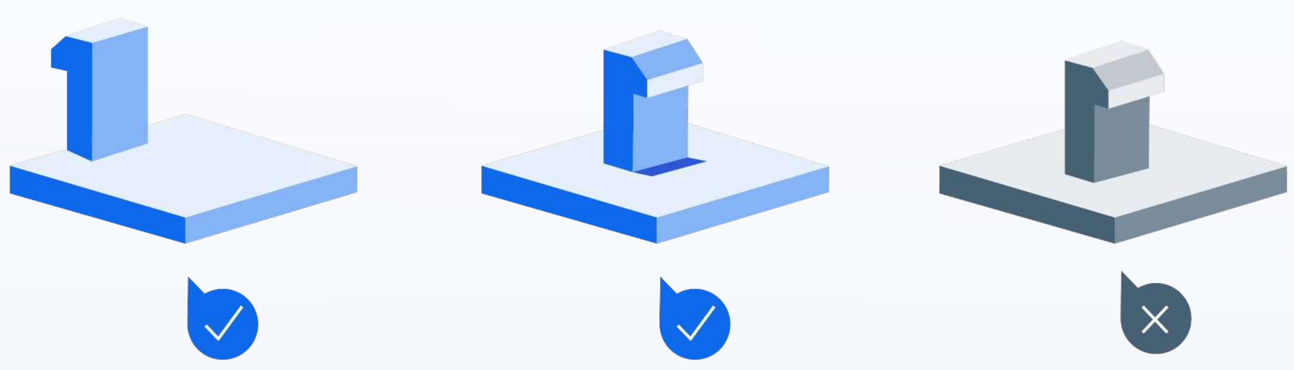
Type of Snap-fit Joints	 Cantilever Snap-fit	 Torsional Snap-fit	 U-shaped Snap-fit	 L-shaped Snap-fit	 Annular Snap-fit
Applications	Consumer electronics, toys, small enclosures	Closures, removable panels, product housings	Packaging, product enclosures, clamp-like mechanisms	Packaging, housing lids	Cosmetic containers, jars, bottle lids, ball-and-socket joints in automotive
Advantages	• Simple • Cost-effective • Easy disassembly/ reassembly • Versatile with various materials	• Easy disassembly without damaging parts	• Increased flexibility • Quicker assembly, • Less stringent tolerances	• Excellent lateral holding power • Ideal for preventing disengagement from side impacts	• Uniform engagement distributes stress evenly • Ideal for high-load, liquid/air-tight applications
Challenges	• Stress concentration at the base can lead to fatigue • Use of fillets can reduce stress	• Repeated torsional loads can cause wear • Materials like nylon are preferred	• Rigid materials can cause wear; polypropylene or TPE helps	• Difficult to design for disassembly • Material selection crucial for side load resistance	• Requires precise manufacturing tolerances • Difficult disassembly
Load Capacity	Low to moderate	Low to moderate	Low to moderate	Moderate	High (360°)
Reusability	High	Low	High	Moderate	Moderate

Snap-Fits

General Design Guidelines for Snap-Fits

When designing snap-fits, many important considerations are involved, including material, dimensions, and pass-thru core design. The flexibility of plastic materials is one of the most important features that makes snap-fits functional. Undercuts that may prevent the movement of the mold and create a shadow on the part should not be made. Here are our top general guidelines for snap-fits:

- Avoid sharp corners at the base of cantilevers
  - Radii should be at least half the thickness of the cantilever base and no smaller than 0.38 mm (.015”).
- Taper the thickness of cantilevers to avoid uneven stress distribution and lower material usage.
  - It is recommended to reduce the thickness linearly so the tip is half the base thickness.
- Aim for a tip width of 5mm or greater for increased strength.
- Ensure to design for an appropriate gap between the mating surfaces:
  - 0.2 mm for tight fits
  - 0.3 mm for close fits
  - 0.4 mm for slide fits or pivot joints
- If possible, move cantilever snaps to the outside edge of the part to avoid undercuts and complex tooling, or design a slot under the hook overhang.



Snap-Fits

Calculating Ideal Dimensions for Snap-Fit Designs

To design the ideal snap-fit, you will need to understand and consider the interplay between material properties, geometry, and the forces that will be involved. By considering the necessary factors, it's possible to calculate and verify suitable snap-fit design parameters. In the section below, we will run through an example of using standard beam theory formulas to guide a rectangular cantilever snap-fit design.

The first key factor to consider is the material you'll be working with. That will dictate the Elastic Modulus (E) and Maximum Strain (ε) values that will be used in the calculations. The next factors to keep in mind are the geometric parameters, such as the Length (L), Thickness (t), and Width (w). Next, you should determine how much Deflection (δ) there will be, as well as the Force (F) required to engage the snap-fit.

From here, you can use deflection and strain formulas to determine ideal parameters and verify that they are within safe limits for the chosen material. In our example we want to find the ideal beam thickness of a rectangular cantilever snap-fit made of ABS. The formulas we will use in our calculations are shown below:

Deflection (δ):

$$\delta = \frac{FL^3}{3Ewt^3}$$

Strain (ε):

$$\varepsilon = \frac{6\delta t}{L^2}$$

Snap-Fits

Calculating Ideal Dimensions for Snap-Fit Designs

For our example, we will assume the following:

- A deflection of 2 mm (**δ = 2 mm**)
- An applied force of 5 newtons (**F = 5N**)
- An Elastic Modulus (E) of 2.4GPa (**2400 MPa**)
- A Maximum Tensile Stress (σ) of **34 MPa**
- A cantilever length of 30 mm (L = **30 mm)**
- A cantilever width of 10 mm (w = **10 mm**)

We want to find the thickness (t) of the cantilever that satisfies the strain limit. To do so, we will rearrange the deflection formula like so:

$$t = \left(\frac{FL^3}{3Ew\delta} \right)^{1/3}$$

Next we will substitute the values and normalize our units in the formula to SI base units:

- F = 5 N
- L = 30 mm = 30×10^{-3} m
- E = 2400 MPA = 2400×10^{6} Pa
- w = 10 mm = 10×10^{-3} m
- δ = 2 mm = 2×10^{-3} m

$$t = \left(\frac{5 \times (30 \times 10^{-3})^3}{3 \times (2400 \times 10^{6}) \times (10 \times 10^{-3}) \times (2 \times 10^{-3})} \right)^{1/3}$$

Snap-Fits

Calculating Ideal Dimensions for Snap-Fit Designs

When we convert the result back to millimeters, we get a thickness (t) of approximately **0.979 mm**.

Now, let's verify that the design does not exceed the strain limit by using the strain formula, where:

- ➔ $\delta = 2\ mm = 2 \times 10^{-3}\ m$
- ➔ $t = 0.979\ mm = 9.79 \times 10^{-4}\ m$
- ➔ $L = 30\ mm = 3 \times 10^{-3}\ m$

This gives us a calculated strain of approximately **0.0130 (13.05 mε)**. Now, we just compare this to the maximum strain (εmax) of our material, which can be calculated like so:

$\varepsilon max = \sigma/E$

$\varepsilon max = 34/2400 = 0.0142\ (14.2\ m\varepsilon)$

Since our design's calculated strain is 13.05 mε and below the material's maximum strain of 14.2 mε, it means our design is within the safe limit. By dividing the maximum strain by the calculated strain, we get a safety factor of approximately 1.088. For general applications with relatively predictable loads and conditions, we recommend aiming for a safety factor of **1.5 to 2**. In critical applications where failure could result in serious consequences, such as in medical devices or safety-critical components, a safety factor of **2.5 to 3** is recommended.

Texts & Symbols

You may want to add details such as text, logos, recycling information, or patent information to your product. Common stamps such as date and recyclability are standardized and easily applied via molding inserts; these can be easily requested during quoting within Xometrys quoting engine.

Text and symbols can be embossed or engraved directly onto the mold, which will automatically transfer to the molded parts. This eliminates the use of stick-on labels or other part marking post-processes, which can add cost. However, some considerations must be made with text and logo features to ensure an optimal design.

Guidelines For Embossed and Engraved Features

Embossed (raised) text on the part requires engraved text on the mold and vice versa. Embossed text on your part design is recommended as it is easier and quicker to apply the corresponding engraving onto the mold through CNC machining. Raised text on the molded part is more durable and highly visible. The characters often stand out and are easily readable from a distance; another reason why embossed text in your molded part design is recommended.

Texts & Symbols

Guidelines For Embossed and Engraved Features

On the other hand, engraved text on parts means the corresponding area of the mold needs to be embossed, which appears as a raised surface surface on the mold. Embossing the mold is a more demanding and time-consuming process, often requiring more specialized tooling or techniques, which increases production costs.

This approach is also associated with accelerated tool wear and stricter quality control requirements. These factors can further increase production costs, with the overall impact depending on the design complexity and production volume.

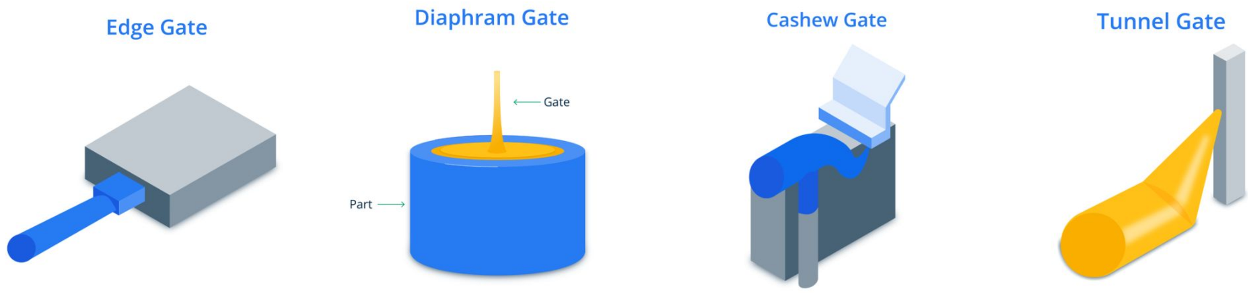
Recommendations for Visibility and Readability

These guidelines and tips will help ensure you get the most visibility and readable text and logos on your molded parts:

- ➔ Consider embossed text with a **height of 0.5 mm**.
- ➔ Consider a uniform thickness of the lettering and a **font size of 20 points**.
- ➔ **Sans-serif fonts** are generally preferable due to their balanced character width.
- ➔ Ensure the letters **align perpendicularly to the parting lines** or line of draw.

Gate Design

Any injection molding mold design is incomplete without gates. These components are the entry sections through which the resin flows into the cavity. They control the flow of the molten plastic and directly influence the quality of the molded parts.



The size, shape, and location of these gates significantly impact the end products in terms of appearance and structural integrity. They should be considered during the mold design to ensure they are not the cause of defects such as flow marks and weld lines.

Choosing Gate Positions to Minimize Defects

A gate's placement depends on the part geometry, mold design, and molding material. An ideal gate location helps eliminate secondary de-gating operations which can be time-consuming and add additional challenges. We recommend placing gates away from high-stress or impact areas.

The general rule is to place gates in a position where the cavity is effectively filled and usually in the thickest area. In some cases, where the mold design is complex, multiple gates may be necessary. Consider this if the part is relatively large and has complex geometry.

Gate Design

Gates often leave plastic protrusions from the part's surface, which require trimming after the part is ejected from the mold. However, this is not the most ideal approach, as the marks left behind create visible surface imperfections. Placing gates along the mold parting lines can help alleviate this.

Common Gate Types

Xometry's engineers and molders can help you decide the right gate type and location based on your product design during the DFM review process. That said, it is good to familiarize yourself with the most common types of gates and their applications.

Edge Gate: As the name implies, this gate type allows resin injection into the mold cavity at the parting line of the two mold halves. It is, by far, the most common gate type because it only leaves a tiny surface imperfection after the system has been removed.

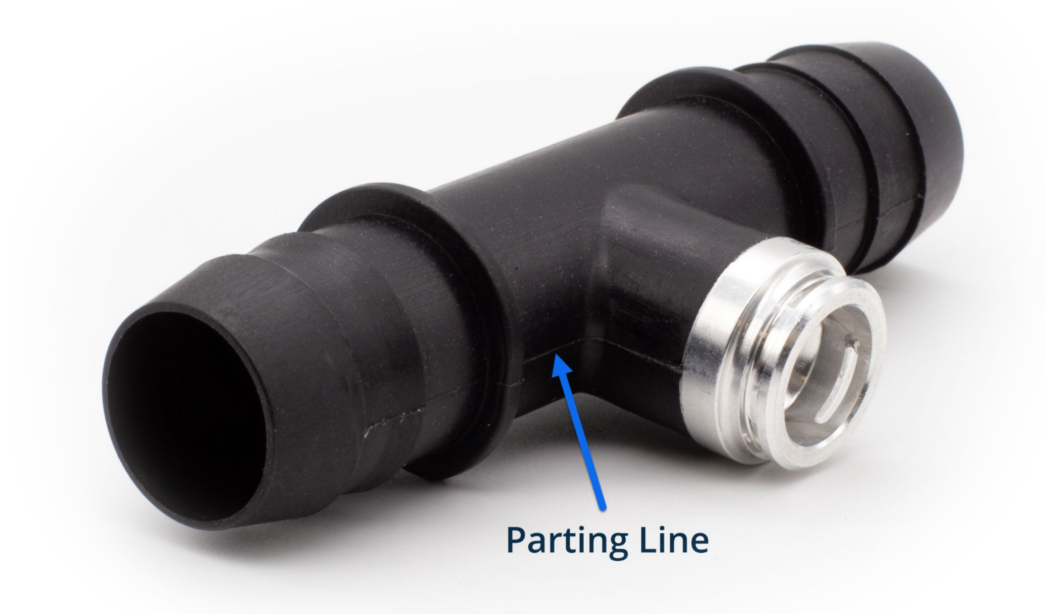
Hot Tip Gate: A type of gate directly connected to the spur to allow the molten resin to be injected from the top side of the part. This gate is preferred for mass production of parts because it is more efficient. It, however, leaves a visible mark at the injection point.

Fan Gate: Known for its relatively large opening characterized by variable thickness. As a result, fan gates are ideal when designing molds for large parts, as molten resin under pressure can easily be forced out through the gate.

Pin Gate: This gate differs from other gates in that it permits high injection speeds. Pin gates are the ideal choice for molding consumer products as they enhance aesthetic appeal. Consider it if you don't want vestiges to appear on any side of the parting line.

Parting Line

Parting lines are usually visible on the side of the part where two halves of the molded parts meet. The causes are the misalignments of the molds and rounding of the mold edges during the design process.



Making the parting lines as simple as possible helps in making cleaner molded parts more quickly. It can be challenging for any designer to make the parting lines unnoticeable on injection molded components, especially while trying to maintain aesthetic appeal. You have to decide on their precise location without compromising the functionality of the end products or introducing defects such as flash.

Parting Line

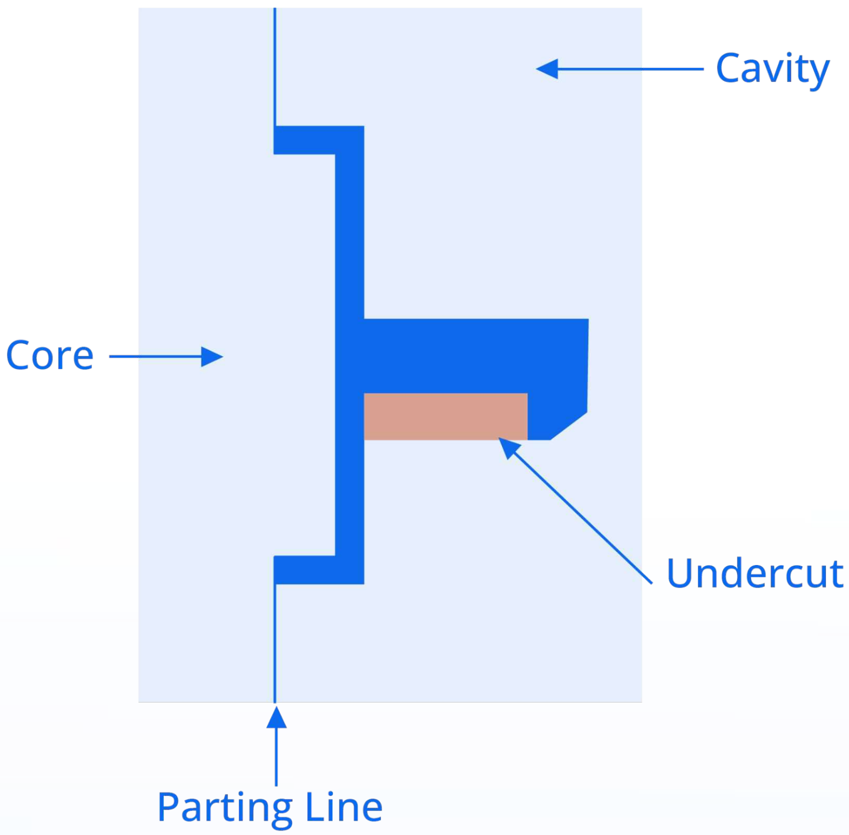
Tips for Strategic Parting Line Placement

Parting line placement heavily depends on the part's geometry and function. It is always visible on the end product, but its location can be optimized to lower visibility while maintaining the part's functionality. For example, many designers strategically position parting lines along feature edges—such as rims, buttons, and caps—to minimize their visual impact. This placement also facilitates smoother mold release once the resin has cooled and solidified.

Avoid placing parting lines on critical function areas. As mentioned, parting lines are where two halves of the mold meet. There may be minor consistency changes between runs, flash, and variations that can affect performance.. The best approach is balancing aesthetics with functionality. Strategic and well-placed symmetrical parting lines ensure they do not detract from the part's overall design.

Dealing With Undercuts

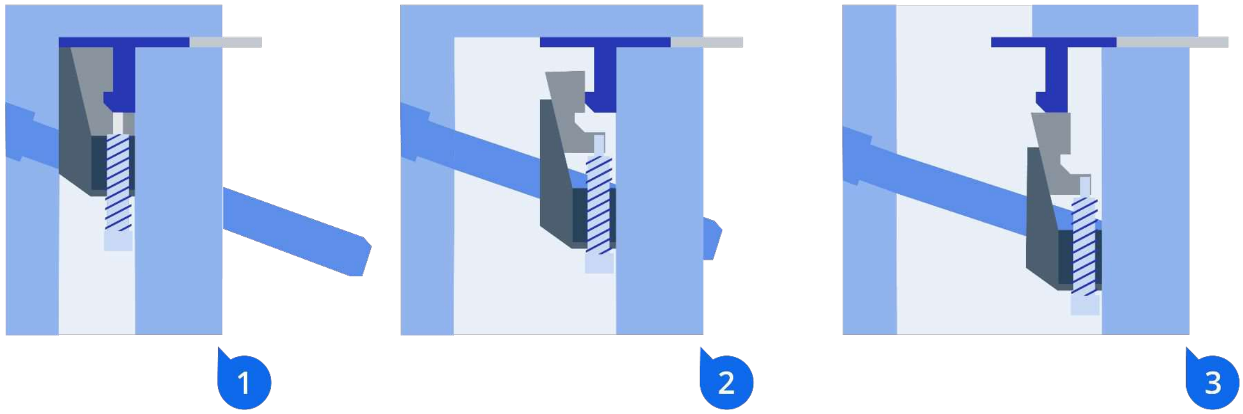
Undercuts are features such as cavities, protrusions, threads, or recessed areas that prevent a part from easily ejecting from a mold. While many plastic parts can be molded using a simple two-part mold, undercuts require additional mechanisms like side cores, lifters, or sliders to enable proper release.



When undercuts appear on the exterior of a part, side cores slide in and out with each molding cycle.

Dealing With Undercuts

Undercut Lift Mechanism



If multiple undercuts exist on the same side, a single side core can sometimes accommodate them. Internal undercuts, on the other hand, are managed by lifters, which operate similarly to side cores but work inside the mold. However, both side cores and lifters have depth limitations due to mold tooling and press constraints.

Because undercuts add complexity and increase tooling and production costs, designers should aim to minimize them whenever possible.

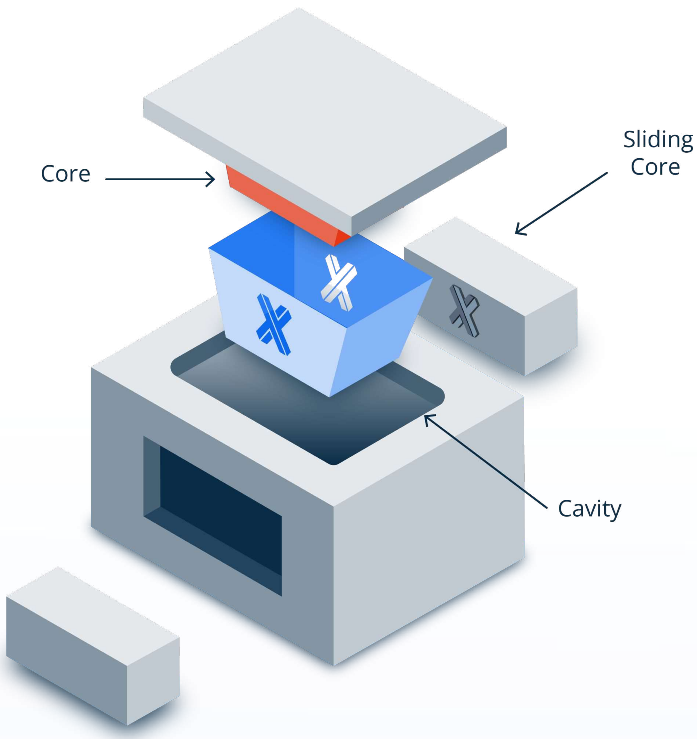
Clever design alternatives—such as pass-through cores—can sometimes eliminate the need for side cores, provided the part is flexible enough for easy ejection. By optimizing mold design early, manufacturers can reduce costs while maintaining functional part features.

Dealing With Undercuts

Sliding Cores

Sliding cores are movable inserts incorporated into a mold to form undercut features and facilitate part ejection. Features such as cuts or holes with hollow bottoms can create undercuts that prevent a part from being removed using a simple two-plate mold. By moving laterally, or at an angle relative to the parting line, during the mold's opening and closing cycle, sliding cores allow these complex features to be molded and then safely retracted to release the finished part.

Key points to consider when designing with sliding cores include:



- **Positioning**: Sliding cores can be arranged parallel to or at an angle relative to the parting line, depending on the part's design.
- **Clearance**: Sufficient space must be provided within the mold to allow the cores to move freely.
- **Draft Angles**: A draft angle of at least 1° should be incorporated on the surfaces where the sliding cores contact the molten plastic. This aids in the smooth ejection of the part and reduces the risk of damage.
- **Cost Considerations**: Because the use of sliding cores adds complexity and increases tooling costs, their inclusion should be minimized through smart design whenever possible.

Dealing With Undercuts

Shut-Off

In many cases, the most effective way to address problematic undercuts is to eliminate the troublesome area altogether. A shut-off is a design feature built into the mold where the two mold halves come together. At this meeting point, the shut-off creates a barrier that prevents the plastic from flowing into certain areas. This means that when the mold closes, the plastic simply doesn't fill the shut-off zone, eliminating the undercut without the need for extra moving parts like sliding cores. This approach provides a low-cost alternative to using sliding cores, which add extra tooling costs.

Shut-offs are especially effective for handling undercuts with hollow bottoms or those along the edges of a part. When designing shut-offs, it’s important to include a draft angle of **at least 3°-5°** on the surfaces where the plastic meets the shut-off. This not only ensures a secure shut-off but also helps prevent issues like cavity formation and reduces wear on the mold.

Additionally, by combining shut-offs with features such as pass-through cores, you can strategically form detailed features like open and closed holes, hooks, and cams that move laterally. This integrated approach simplifies the mold design while still achieving the desired complex part features.

Moving the Parting Line Position

Moving or repositioning the parting line (the interface where the two mold halves meet) can effectively eliminate undercuts, especially on the outer surface of a part. By adjusting the parting line, you prevent the plastic from filling areas that would otherwise form undercuts.

Dealing With Undercuts

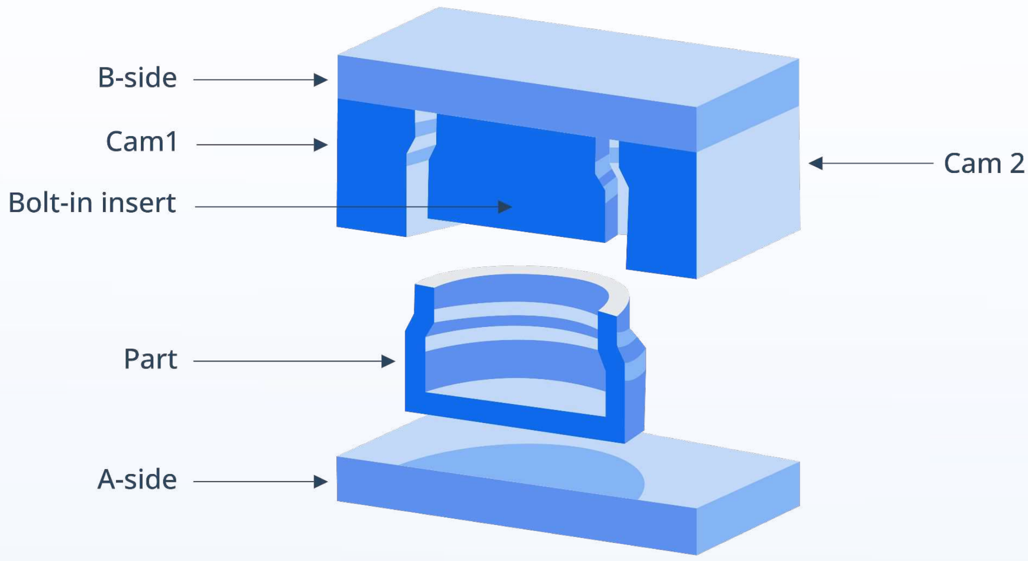
This solution not only avoids the need for additional moving parts like sliding cores or lifters but also simplifies the mold design, reduces tool costs, and ensures the ejection mechanism works smoothly. However, moving the parting line may not be suitable for all part designs, especially those with more complex geometries.

Bump-Offs

Bump-offs are small, removable attachments built into the mold that help release a part by breaking the hold created by undercuts. As the mold opens, these bump-offs move with the ejection process, providing a localized push that "bumps" the part free. This action eliminates the need for more complex components like sliding cores and is especially effective for parts made from flexible, unfilled materials such as TPE, PE, or PP, which can tolerate slight deformation during ejection.

For optimal performance, bump-offs should be integrated with proper draft angles and positioned away from support elements like corners and ribs. They work best in designs with hollow structures and undercut features, where they minimize sticking and reduce the risk of damage during demolding.

However, bump-offs are less suitable for stiffer materials or those reinforced with fillers, such as glass fibers, which tend to snap rather than flex.



Plastic Part Tolerances

In injection molding, the typical design process prioritizes form and function first, where specific tolerances are used to define critical features. That said, it can be helpful to understand the relationship between part tolerances and the process. Injection molds are precision machined from aluminum or steel. These molds are machined to typical CNC tolerances of **± 0.13 mm**, unless a tighter tolerance is specified. When plastic is injected into a mold, it cools and shrinks. The exact amount of shrinkage is a function of the resin being used. Each mold is machined slightly larger than the part to account for shrinkage of the resin when it cools.

Although this shrinkage rate is very predictable, slight variations in the resin affect the shrinkage and hence the final part tolerance. The shrinkage variation gets larger as the part gets larger, so depending on the material you should expect the tolerance due to shrinkage to be roughly **± 0.05 mm/mm**.

For example, a 4” ABS part will have a tolerance of roughly +/- .010-.011” (0.28 mm). Part to part, the repeatability is typically under +/- .004". If your tolerance needs are tighter than standard plastic tolerances, please let us know your design requirements, and we will work with you to meet them.

As with most processes, as tolerance requirements increase, so does the cost. Designers should strive to balance precision with manufacturability to ensure a consistent, cost-effective production.

Plastic Part Tolerances

Some other design considerations to make with setting tolerances are:

- Considering your material choice: some resins have higher shrink rates than others, which can influence ability to hit precise tolerances.
    - Semi-crystalline materials (e.g., Nylon, PP) generally have higher shrinkage than amorphous materials (e.g., ABS, PC).
- Only applying strict tolerances to critical dimensions and allowing looser tolerances where possible.
- Keeping walls as uniform as possible to improve consistency.
- Molds can wear over time, therefore if a part requires long-term production, consider designing tolerances with mold maintenance in mind.

DESIGN GUIDE

Common Defects & How to Prevent Them

Warping

Plastic materials tend to shrink as they cool in the mold. Different areas of the part can cool and shrink at different rates, especially when wall thicknesses are uneven. This leads to distortions and an uneven appearance known as warping.

To prevent excessive warping, any necessary variations in wall thickness should **not exceed 15% of the nominal thickness**.

Additionally, transitions between different thicknesses should be smooth or tapered to promote even cooling and maintain part integrity. The key to avoiding warpage is to design your part so it cools down as uniformly as possible. If our molding experts identify potential issues due to non-uniform wall thickness, we will recommend design modifications to optimize manufacturability and ensure a high-quality final product.

Sink Marks

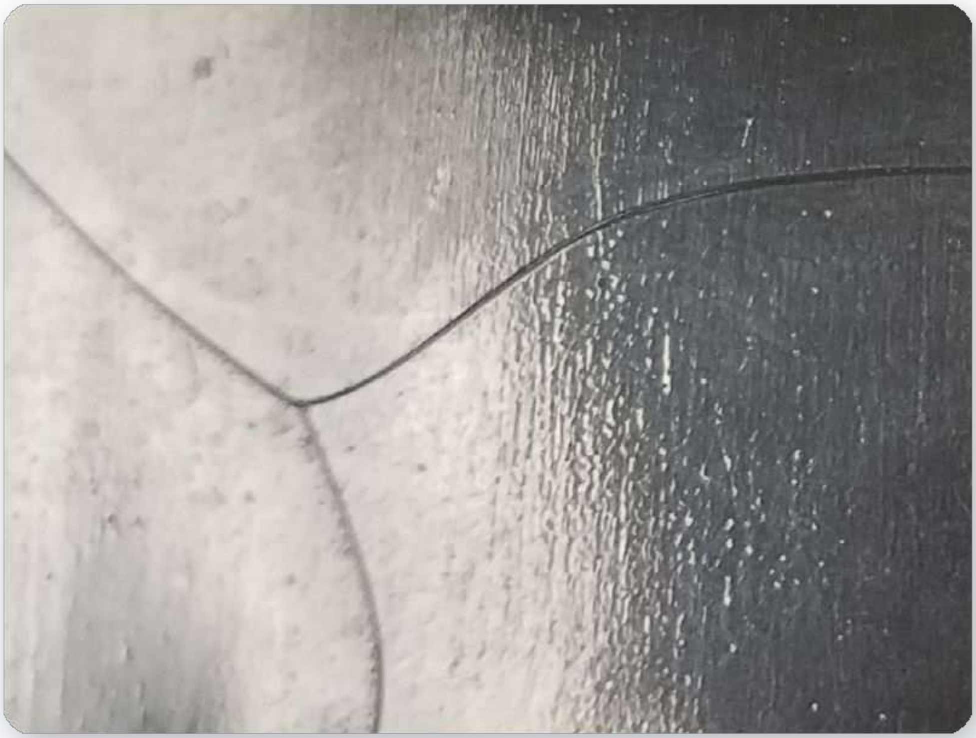
Similar to warping, sink marks are defects that can occur due to non-uniform thicknesses and uneven cooling. As the name implies, these appear as sunken areas or dimples on part surfaces. They typically occur in thicker sections where the outer surface cools and hardens faster than the inner material. As the inner material continues to cool and shrink, it pulls the outer surface inward, creating a visible indentation.

An example of a part with sink mark

Again, the goal should be to create even and uniform thicknesses with smooth transitions to promote uniform cooling. Ribs, hollows, or other design features that can core out excess material can help prevent sink marks from occurring. Some materials have a higher shrinkage rate, making them more prone to sink marks; however, material additives can help mitigate this.

Knit Lines

Knit or weld lines are visible lines or weak points that form in an injection-molded part when two or more flow fronts of molten plastic meet and fail to fully bond. They typically occur when the plastic encounters an obstacle, such as a hole, boss, or insert, causing the flow to split and then rejoin. A weak bond forms if the material cools too much before merging, resulting in a visible line that can reduce the part's structural integrity and visual appeal.



An example of knit lines on a part

To minimize knit lines, designers can adjust gate placement and modify the part geometry to optimize flow paths. Using resins with better flow properties or adding reinforcing materials can also help strengthen knit line areas and improve overall part quality. To ensure the best results, Xometry will determine the optimal gating and knit line locations. If you have specific preferences, we're happy to discuss them with you.

Drag Marks

Drag marks in injection molding are surface defects that occur when a part experiences friction or scraping as it is ejected from the mold. These marks typically appear as scratches, scuffs, or streaks on the part's surface and are most common in areas with vertical walls or insufficient draft angles.

To minimize drag marks in injection-molded parts, designers should focus on reducing friction between the part and the mold during ejection. Key design considerations include:

- **Incorporating Adequate Draft Angles**: A minimum draft angle of 1-3 degrees per side is recommended to allow smooth part release and reduce surface contact with the mold.
- **Optimizing Surface Finish**: Highly textured or rough surfaces increase resistance; using smoother finishes in critical areas can help.
- **Minimizing Undercuts and Sharp Features**: Features that create mechanical resistance should be redesigned or adjusted to allow for easier release.

Splay

Splay is a surface defect in injection molding that appears as streaks or silver-like lines on a part's surface. It is typically caused by moisture, trapped air, or volatile substances in the resin that turn into gas during molding. As the molten plastic flows through the mold, these gas pockets disrupt the material's consistency, resulting in visible streaks or a cloudy appearance.

Minimizing splay is typically the responsibility of the molder as it is essential to address moisture and contamination in the material and optimize processing conditions. Key strategies include:

- **Proper Material Drying:** Many resins, especially hygroscopic ones like nylon and PET, absorb moisture from the air. Thorough drying before molding prevents steam from forming and causing splay.
- **Optimized Injection Speed and Pressure:** If the injection speed is too high, excessive shear heating can create gas pockets. Adjusting speed and pressure helps maintain a consistent material flow.
- **Preventing Material Contamination:** Foreign substances, including degraded resin or improper additives, can cause splay. Using clean, high-quality resin ensures a more uniform appearance.
- **Venting and Degassing:** Proper mold venting allows trapped air and gases to escape, reducing the likelihood of splay defects.

DESIGN GUIDE

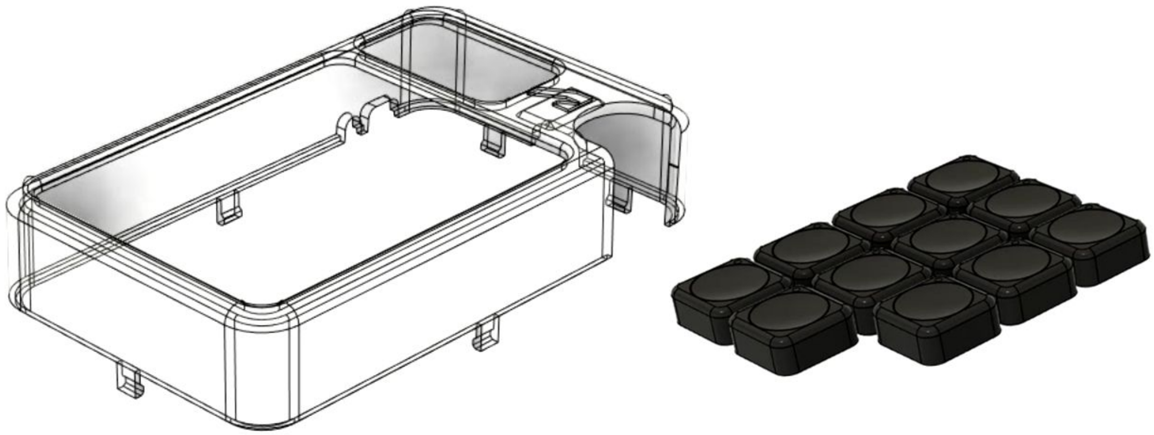
Injection Molding in Action

Autodesk University Use-Case

In this section we'll focus on a real world injection molding project Xometry worked on. Autodesk University is an annual conference held by Autodesk where industry experts, customers and the Autodesk team gather to learn and share knowledge about the latest technologies and trends in manufacturing and product development. During the conference a "Factory Experience" is held which is a hands-on demonstration of the product development process where attendees assemble a product themselves while learning about the components and manufacturing steps involved along the way.

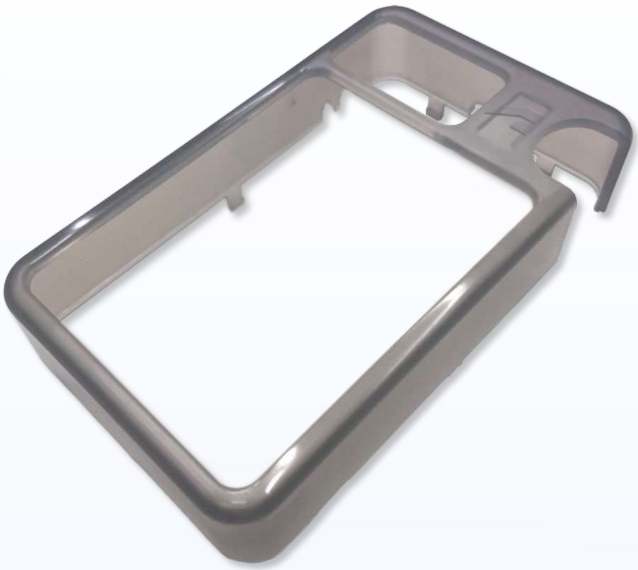
While Xometry has partnered with Autodesk over the past few years to provide components made via a variety of our manufacturing processes for the factory experience, for the purpose of this guide we will focus on how we utilized injection molding for a few parts of the macro keypad pictured to the right.

The Molded Components



We used injection molding to manufacture two types of components on the keypad: the enclosure and the keycaps. Starting with the enclosure, one of the material criteria was it needed to be a rigid and translucent with a smokey grey appearance. As covered earlier in this guide, amorphous thermoplastics can have these properties, so Makrolon 2407, a polycarbonate resin, was chosen.

Color additives were used to customize the coloration. It was important that the exterior surface be moderately polished and free of tool marks as it would be highly visible, so a SPI-B3 finish was specified for those surfaces, while the others would receive a SPI-C3 finish to optimize cost. The image shows one of the molded enclosures:



One of the molded enclosures

The Molded Components

The keycaps were slightly more complicated. One desired characteristic was to be able to engrave different symbols and characters into the caps that would correspond to the keypads functions and allow light to pass through the engravings via a backlight, while maintaining an opaque appearance elsewhere.

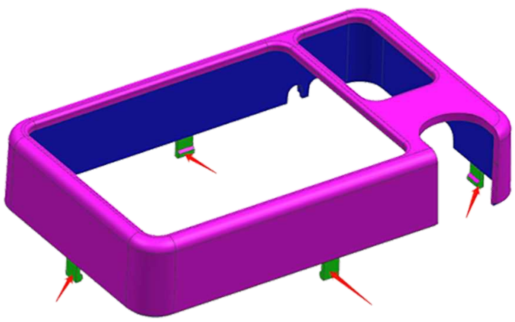
This lead us to utilizing an **overmolding** approach for the keycaps, using translucent Makrolon 2407 for the substrate material with an opaque black polybutylene terephthalate (PBT) overmold on top. This allows light to shine through just the engraved portion of the caps, creating a visually appealing and clean effect. A SPI-B3 finish was chosen for the cavity side of the mold for a medium-polish sheen.

The finished overmolded keycaps complete with engraving

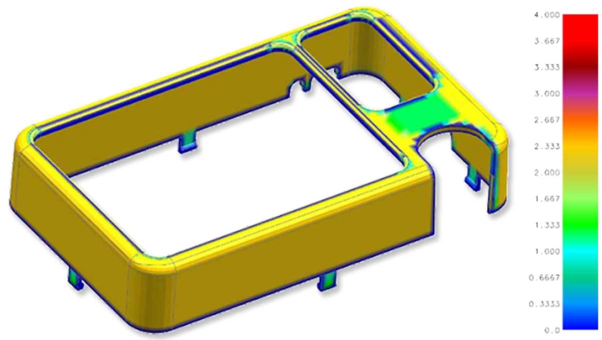
Enclosure Molding Strategy

To mold the enclosure, we start by performing various types of analysis on the part design using software tools that can check for things like undercuts and wall uniformity.

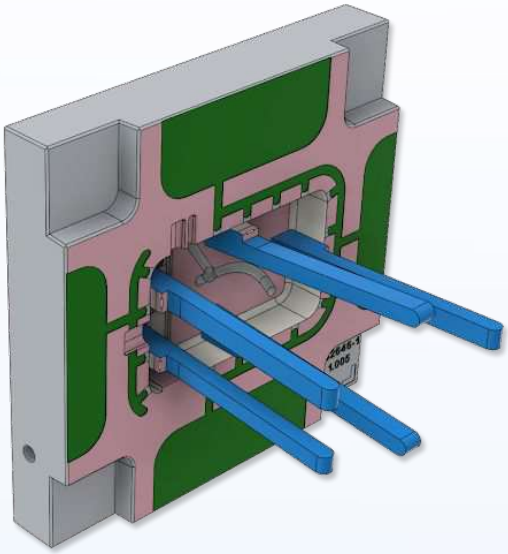
Undercuts



Uniformity



This helps us determine the best tooling strategy when other factors such as quantity, budget and longer term needs are accounted for. In the undercut analysis above (left) you'll notice the tabs on the bottom of the enclosure were flagged. The wall thickness analysis (right) shows that uniformity is relatively consistent across the part.



For this iteration of the design, the clips on the bottom of the enclosure were necessary and the undercuts couldn't be addressed by re-positioning them or implementing pass-throughs. To handle the undercuts during molding, we used lifters into our tool, as seen in blue in the image to the left.

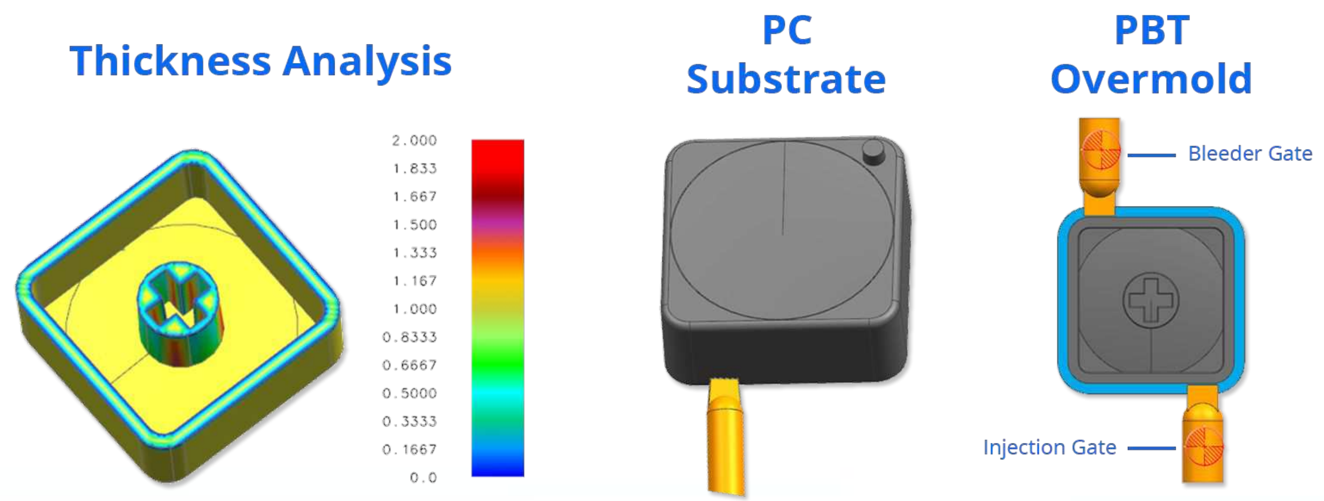
Enclosure Molding Strategy

Maintaining part appearance and functionality is critical. As we covered earlier in this guide, strategically using the right type of gates and optimizing their locations is a large factor of meeting those requirements. For the enclosure design we determined that a sub-gate system with two fan gates would work best.

One gate was placed in the upper left section of the part, while the other gate was placed opposite to it. This ensured the plastic would be able to flow and fill the entire mold evenly without defects. Gates can leave minor vestiges behind where they were connected to the molded part, so we were sure to communicate and get approval from the client for the gate locations.

Keycap Molding Strategy

The molding approach for the keycaps was a bit different due to the overmold process. In some ways, the tool was simpler because the keycaps had no undercuts or wall thickness issues, allowing for a standard 'open and close' type tool that required no lifters or slides. On the other hand, this tool needed to be designed to economically support a small production run of overmolded parts with an accurate mating feature.



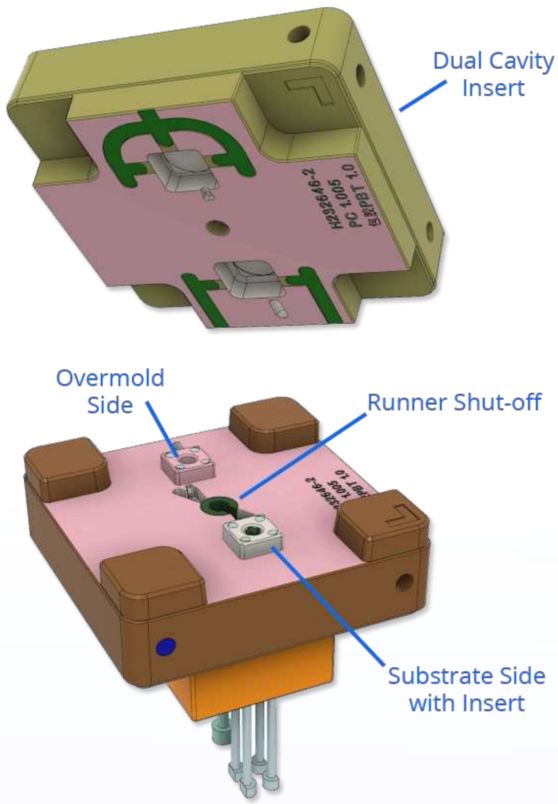
In the image above you can see a cross-shaped feature on the caps bottom side which allows it to mate with the off-the-shelf mechanical switches which would be used on the keypad. A special type of molding insert pin was used to form the feature. In the initial T1 sample run it was determined that the keys were fitting more loosely than desired, but since the insert was 'steel safe' it meant we could respond rapidly by grooming the tool to tighten the fit, which worked great on the subsequent sample run and final parts.

Molding the rest of the caps was straight forward; edge gates were used to mold the polycarbonate substrate (shown in grey in the image above). You'll notice on the PBT overmold shot (shown in blue coloration) there are two gates.

Keycap Molding Strategy

One of these gates is used for injecting the material, where the other was strategically placed as a “bleeder gate”, which helps avoid defects like knit lines and also helps material flow, ensuring the thin section of the overmold was filled entirely. The top of the overmold was designed intentionally thin as this was the surface to be partially engraved away to produce the backlit markings.

To keep cost economical at the quantity of parts needed, we designed the tool as a family tool; containing both the substrate and overmold cavities in the same tool. You may be wondering how two different materials are handled with the same tool. That is where a runner shut-off comes into play. This feature, which you can see in the center of the tool core in the image to the right, allows us to choose which cavity of the tool to use at a given time. Once the shots for the substrate are complete, the material can be purged and the shut-off adjusted to guide the secondary material into the overmold cavity.



One last thing to note is the tool was designed to work with a vertical press, which helps with overmolding or insert molding, as the substrate can simply be held in place by gravity.

Through the combination of good part design, communication and clever tool strategy, we were able to successfully deliver the components for this project within the target timeframe and budget!

DESIGN GUIDE

Injection Molding With Xometry

Starting An Injection Molding Order

Getting an injection molding project started is easy thanks to Xometry! All you need to get started are 3D CAD files of the parts you're looking to get manufactured. With your CAD files in hand, just follow these simple steps to get your injection molding project underway:

1. Upload your 3D CAD design files to the instant quote section of your **personal dashboard**.
2. Once your parts have analyzed, click the '**Edit specifications**' button.
3. Specify the quantity needed in the corresponding field.
4. Specify your target material, additives, finishes and any other selectable features that apply to your project (e.g. inserts).
5. If you don't see an option you're looking for in any of the configuration options, feel free to choose 'Custom' and let us know what you are looking for and we will be happy to review.
6. Click the '**Back to Quote**' button to save your configurations.
7. You can specify inspection and certification requirements from the options shown, if needed.
8. At this stage, you have two options:
    a. If available, you can get an instant quote for your project and click the '**Proceed to Checkout**' button.
    b. If not, you can request a manual quote. Our team typically responds within 24–48 hours.

The Process of Injection Molding with Xometry

Create an online quote at Xometry, and our team of engineers will guide you through each step, from quoting and Design for Manufacturability (DFM) to tool creation and parts delivery.

1. **Submit Your Injection Moulding Enquiry**: Upload your designs, select the quantity, materials, surface finish, and other parameters, then submit a quote request.
2. **Review and Quoting**: A Xometry engineer reviews your enquiry within 24-48 hours to ensure all requirements are met. You receive an initial quote. Upon your approval, our engineers match your project with the most suitable manufacturer based on skills, availability, and other criteria. We then send you a final quote.
3. **Project Kick-Off and DFM**: Once you approve the final quote, we start the Design for Manufacturability (DFM) process. Xometry works with you to clarify and finalise all technical details before proceeding.
4. **Mould Building and Sample Production:**
    a. After finalising DFM, we set deadlines for mould building and first sample production (T1).
    b. You receive an order confirmation and an invoice for 50% of the mould costs.
    c. Our project manager oversees logistics and communication. You receive the T1 samples, along with a measurement report and material certificates for approval.
    d. Feedback on the T1 samples may lead to further iterations (T2, T3, etc.).

The Process of Injection Molding with Xometry

5. **Full Production and Delivery**: Once samples are approved, Xometry manages full-scale production, quality control, and part delivery. The mould will be stored at production facilities for two years, and you won't charged for it when placing repeat orders of the same parts.
6. **Mould Ownership and Re-Orders**: The mould lifespan is between 10,000 to 1,000,000 shots. The mould is owned by you and maintained by Xometry.

Xometry's Injection Molding Standards

Xometry's injection molding standards are optimized for our platform and based upon common industry standards. They are as follows:

- Typical mould machined tolerances are ±0.127 mm.
- Tighter tolerances can be requested and may increase the cost of tooling. Additionally, many tight tolerances require the mould to be manufactured, sampled, and groomed.
- Part-to-part repeatability is typically under ±0.1 mm.
- Lead time stated is for the first T sampling shipment. The remaining production time is confirmed after T sampling approval.
- Typical first article shipments are 5 pieces but may vary based on size, origin, and material.
- Xometry cannot guarantee a perfect colour match per Pantone / RAL or any other colour system. If an exact colour match is required, a customer must provide a Colour sample to Xometry by sending a flat piece of the targeted colour on the Quotation stage.
- All quotes are based upon the assumption that designs have an adequate draft, radii, and coring for manufacturability.
- Cores, side actions, and tooling strategy are determined by Xometry unless explicitly discussed.
- Gating, ejection, knit lines and parting lines are at the discretion of Xometry unless explicitly discussed.

See more of Xometry's Manufacturing Standards

Other Molding Services

Standard plastic injection molding isn't the only type of molding process Xometry offers. Below you'll find related molding services we can assist you with and some quick tips for them:

Overmolding

In basic terms, the overmolding process lets you combine multiple materials into one part. One material, usually a thermoplastic elastomer (TPE/TPV), is molded onto a second material, which is often a rigid plastic. Think about your toothbrush handle where the single piece has both rigid and rubbery components. It's a great way to make plastic parts perform and look better.

Tips for Overmolding:

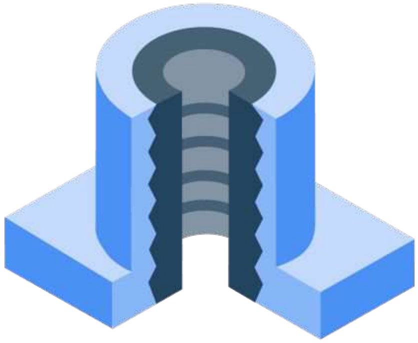
- Wall thicknesses between 1.5 mm-3 mm generally provide the best bonding.
- Keeping radii between 0.5 mm minimum in corners reduces localized stresses.
- If the part requires the use of thick TPE sections, they should be cored out to minimize shrinkage problems, reduce the part weight and lower cycle time.
- Avoid deep or un-ventable blind pockets or ribs in your design.
- Use gradual transitions between wall thickness to reduce or avoid problems with flow (back fills, gas traps, etc.)
- The TPE/TPV should be less thick than the substrate to prevent warpage, especially if the part is flat, long, or both.
- Overmolding needs mechanical or chemical bonding to the substrate, so your overmold material choices should enable this.

Other Molding Services

Insert Molding

Insert molding is an injection molding process where pre-formed components, such as metal or plastic parts, are placed into a mold cavity before injecting plastic around them.

The injected material bonds to the insert, creating a single, solid part. This method is commonly used to add strength, electrical conductivity, or specific features to a plastic part. Insert molding works especially well for parts that have threaded holes. It can also help you create better wheels, pulleys, fan blades, and other similar parts.



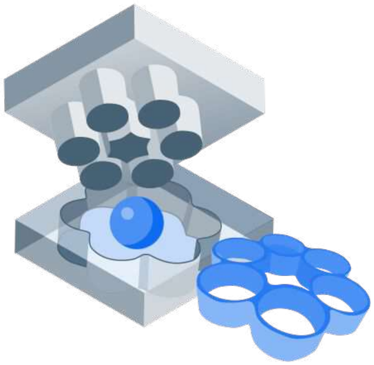
Insert molding can help:

- **Eliminate assembly**: You can make insert molded parts all at once instead of having to do assembly after the fact.
- **Reduce costs**: Insert molding costs more than standard injection molding, but the costs are usually offset by the fact that you no longer need to do post-assembly.
- **Lower the size and weight of the part**: A plastic part with a metal insert is generally lighter than an all-metal part.
- **Improve reliability**: Metal mold inserts will guarantee the function of threads and mitigate wear and tear over the part's life.
- **Improve part strength**: Because it's a "one-shot" process, it can produce stronger parts than overmolding and other processes.

Other Molding Services

Compression Molding

Compression molding is a manufacturing process used to create plastic and composite parts. Through a mixture of heat and high pressure, it squeezes materials—like thermosetting polymers or thermoplastic compounds—into set shapes. It's the heat and pressure that allows solid materials to soften and reform into new structures that are wholly even and cured.



During the process, curing triggers a chemical reaction, which helps give the final product strength and durability.

Compression molding is an excellent option for rubber products that have thicker or uneven walls such as shoe insoles, phone cases, or silicone kitchen gadgets.

DESIGN GUIDE

Additional Resources

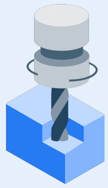
Online Resources

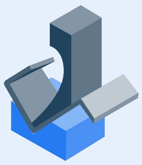
- ➔ **Injection Molding Services Page**
- ➔ **Compression Molding Services Page**

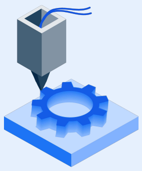
Additional Services at Xometry:

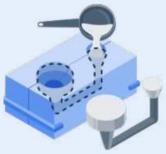
- ➔ Instant Online Quoting
- ➔ **Web:** Upload your CAD file at **get.xometry.eu**
  - **Accepted file types:** STEP (.step, .stp), SOLIDWORKS (.sldprt), Parasolid (.x_t, .x_b), Autodesk Inventor (.ipt), Dassault Systems (.3dxml, .catpart), PTC, Siemens (.prt, .jt), ACIS (.sat), and more!

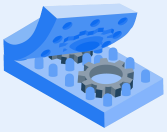
Other Xometry Capabilities:

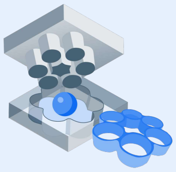

CNC Machining


Sheet Metal Fabrication

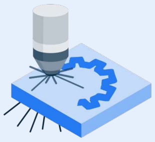

3D Printing


Die Casting


Vacuum Casting


Compression Molding


Laser Cutting


Plasma Cutting

Live Support

Hours: Monday-Friday, 8 AM – 6 PM (CET)
Email: info@xometry.eu
Phone: +49 89-3803-4818

WHERE **BIG IDEAS** ARE **BUILT**