Injection Mold Design Guidelines

Engineering Best Practices for Scalable, Manufacturable Plastic Parts
I.M. Bohannon Mechanical Design

Foreword

Injection molding remains one of the most powerful tools available to product designers and engineers. It enables complex geometries, consistent repeatability, and low per-part costs at scale. But it is also a process that demands precision, where small design decisions made early can have exponential impacts on cost, manufacturability, quality, and tooling longevity. The gap between a manufacturable part and a costly mistake is often measured in fractions of a millimeter.

This guideline was created to bridge that gap.

Whether you're a junior engineer, product designer, engineering manager, or startup founder, this document is designed to provide you with clear, actionable knowledge to design better injection molded parts. It distills years of tooling experience and hands-on production feedback into a format that supports both quick-reference needs and deeper design understanding. It covers fundamentals such as draft angles and wall thickness, dives into mold components like slides and lifters, and outlines essential practices for tolerancing, DFM, material selection, and more.

At I.M. Bohannon Mechanical Design, we work closely with clients at every stage of product development, from concept validation and prototyping to production tooling and supplier engagement. Our goal is to design parts that not only meet performance and aesthetic goals but also simplify manufacturing, reduce tooling complexity, and deliver reliable results under real-world constraints. This guide reflects that approach: practical, precise, and grounded in experience.

Use this document as a reference, a teaching tool, and a conversation starter. If you find yourself wondering whether a boss is too thick, a draft angle is too shallow, or a feature requires a slide, this guide has been written to help you answer those questions. And when you're ready to take a concept further, we're here to help you move forward with clarity and confidence.

Welcome to better part design.

I.M. Bohannon Mechanical Design

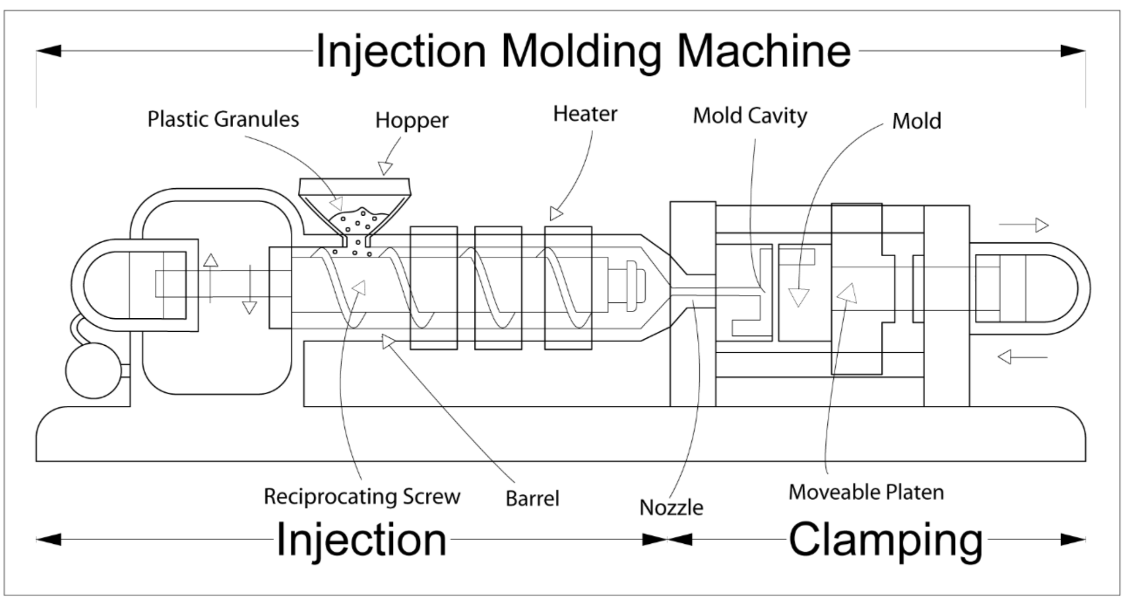
I.M. BOHANNON
MECHANICAL DESIGN

TABLE OF CONTENTS

1. BASICS OF INJECTION MOLDING

Injection molding is a highly efficient manufacturing process used to produce complex plastic parts in large volumes. The process involves melting plastic pellets (typically thermoplastics) and injecting the molten material under high pressure into a precisely machined mold cavity. Once the plastic cools and solidifies, the mold opens, and the finished part is ejected. This cycle can repeat in a matter of seconds, making injection molding one of the most scalable and cost-effective production methods for plastic components.

An injection molding machine consists of three main components: the injection unit, the clamping unit, and the mold itself. The injection unit is responsible for heating and delivering the molten plastic into the mold. The clamping unit holds the mold closed under significant pressure during injection and cooling. The mold contains the cavity (which shapes the outside of the part) and the core (which shapes the internal features), and is typically made of hardened steel or aluminum to withstand thousands or millions of cycles.



The molding cycle begins when plastic pellets are fed into the heated barrel of the injection unit, where they are melted and mixed by a rotating screw. Once the correct volume of material is ready, it is rapidly injected into the mold cavity through a gate. The mold remains clamped shut while the material cools and solidifies. After a set cooling time, the mold opens, and ejector pins push the part out. The mold then closes, and the cycle repeats. Each cycle can take anywhere from a few seconds to a couple of minutes depending on part size, material, and mold complexity.

Injection molding is the preferred method for producing high-volume, repeatable plastic parts with tight tolerances and fine surface finishes. It allows for complex geometries that would be difficult or impossible to achieve with other processes, and it supports a wide range of materials, including flexible, rigid, transparent, and filled plastics. Although the upfront cost of tooling is high, the per-part cost is extremely low, making it economical for mass production. It is widely used in industries such as automotive, consumer electronics, medical devices, and packaging.

2. PARTS OF AN INJECTION MOLD

An injection mold consists of a precisely engineered set of components that shape molten plastic into finished parts. Understanding the function of each component, and the terminology used in design reviews and tooling discussions, is essential for effective collaboration with mold makers and successful part development.

Cavity

The cavity is the A-side (or cosmetic side) of the mold, forming the external surface of the part. It typically includes features visible to the end user, such as texturing, logos, or branding. Cavities are often vented to allow trapped air to escape during fill.

Core

The core is the B-side (or structural side) of the mold. It forms internal surfaces and features such as bosses, holes, and ribs. The core often contains more complex geometry and carries ejector pins for part removal.

Draft Angle

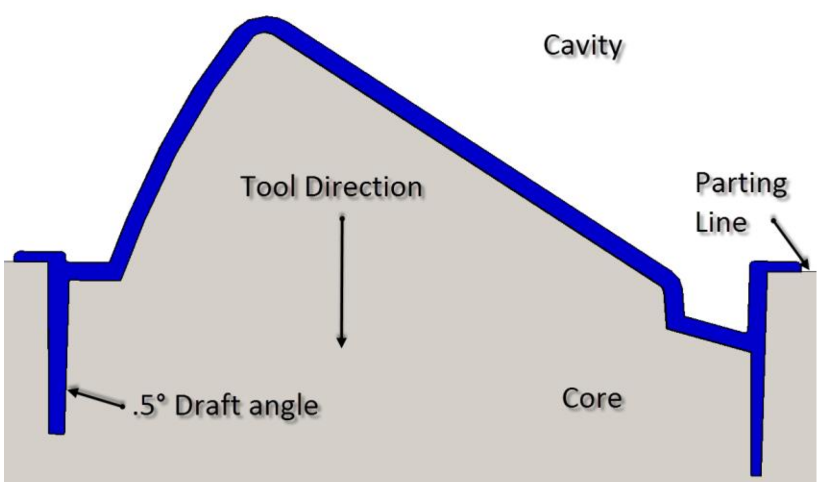
Draft is the slight taper applied to vertical surfaces of a molded part to facilitate easy ejection from the mold. Without sufficient draft, parts may stick or drag, damaging the surface finish or mold itself. Draft angles typically range from 0.5 degrees to 3 degrees, depending on material and surface texture.

Parting Line

The parting line is the visible boundary on the molded part where the cavity and core meet. It defines where the mold separates to release the part. Ideally, parting lines are placed in low-visibility or non-critical areas to avoid cosmetic issues or flash.

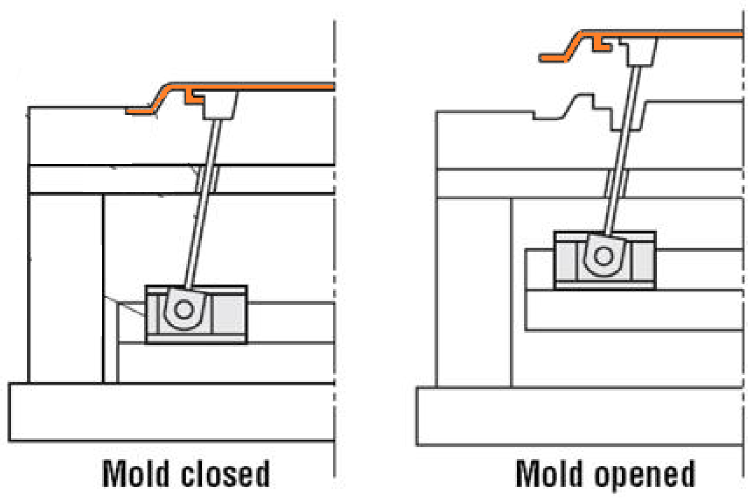
Tool Direction

Tool direction refers to the axis along which the mold opens and closes. All features on a part must be accessible from this direction unless side actions (such as slides or lifters) are used to form undercuts or hidden geometry.



Slides and Lifters

Slides are mechanical components that move laterally (perpendicular to tool direction) during mold opening to form features like side holes, hooks, or windows. Lifters move at an angle to release undercuts or angled features. While both enable more complex part geometry, they require precise timing and alignment during the molding cycle.



Mold Texture

Textures are surface finishes applied to the mold cavity, such as matte, gloss, or grain patterns. They serve cosmetic or functional purposes, such as improving grip or hiding imperfections. Deeper textures require greater draft and can affect mold release.

3. MATERIALS

Injection molding supports a broad range of thermoplastic materials, each with distinct properties that influence part performance, mold design, and processing behavior. Choosing the right material involves balancing factors such as strength, flexibility, chemical resistance, temperature tolerance, appearance, cost, and regulatory compliance. Early collaboration between design and manufacturing teams ensures that material decisions align with functional requirements and production goals.

3.1. POLYMER CLASSIFICATIONS

All plastics used in injection molding fall into two broad categories: thermoplastics and thermosets.

Thermoplastics are the most common in molding. They soften when heated and harden when cooled, allowing them to be re-melted and reshaped repeatedly. This makes them ideal for recycling, prototyping, and mass production. Most commodity and engineering plastics (e.g., ABS, PP, PC, PA) are thermoplastics.

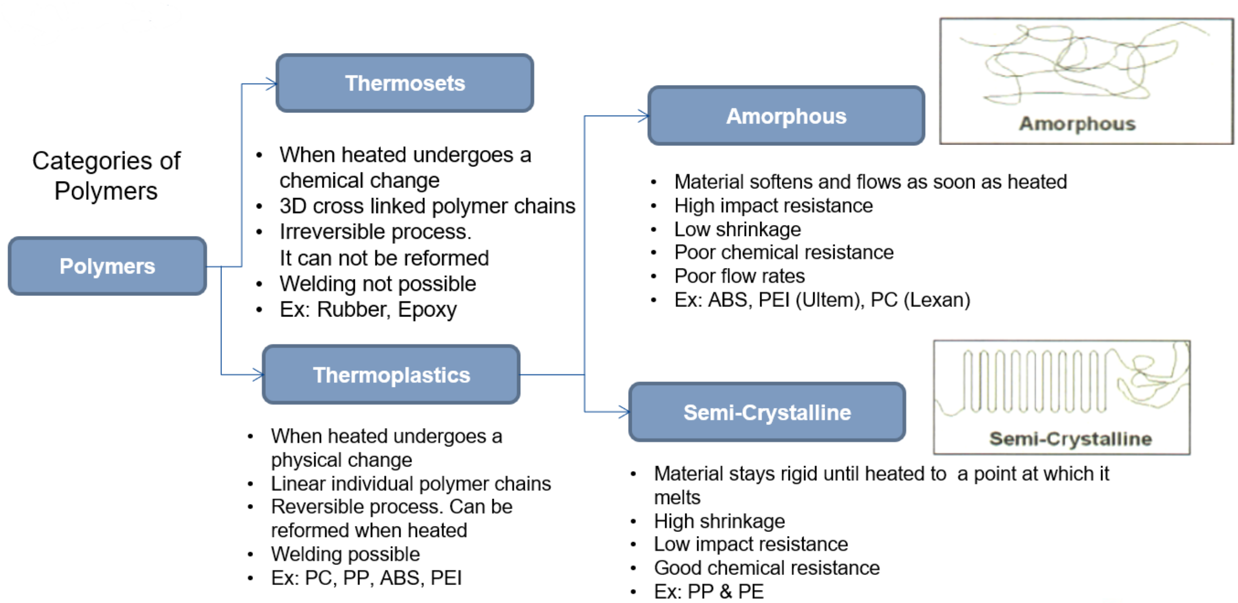
Thermosets, by contrast, undergo an irreversible chemical reaction during curing. Once hardened, they cannot be re-melted. They offer superior heat resistance and dimensional stability but are less common in standard injection molding due to more complex processing. Thermosets are used in high-heat or electrical applications (e.g., epoxy, phenolic, melamine).

Thermoplastics are further divided into amorphous and semi-crystalline materials based on their molecular structure.

Amorphous plastics have a random, entangled molecular arrangement. They exhibit gradual softening, good dimensional stability, and lower shrinkage. These materials are generally clear (e.g., PC, PMMA, PS) and bond well with adhesives or solvents. They are easier to mold and better suited for parts requiring tight tolerances and precision.

Semi-crystalline plastics have regions of ordered molecular chains that form crystals during cooling. They exhibit sharp melting points and are typically more chemically resistant, wear-resistant, and stronger than amorphous materials. However, they have higher shrinkage rates and are more prone to warpage. Examples include Nylon, POM, and PE.

Property	Amorphous	Semi-Crystalline
Shrinkage	Low	High
Clarity	Transparent (e.g., PC, PMMA)	Opaque (e.g., PA, POM)
Moldability	Lower flow rate, stable cooling	Crystalline regions require precise control
Chemical Resistance	Moderate	High
Common Examples	ABS, PC, PS, PMMA	PA, POM, PE, PP



3.2. COMMONLY USED MATERIALS

ABS (Acrylonitrile Butadiene Styrene)

ABS is a tough, impact-resistant thermoplastic known for its strength, dimensional stability, and ease of molding. It has a matte finish and accepts colorants well, making it ideal for both structural and aesthetic applications. ABS balances rigidity and toughness, making it a favorite for enclosures, housings, and consumer products such as automotive interiors, power tool cases, and LEGO bricks. It performs well in moderate temperatures but is not suitable for continuous high-heat environments.

Polypropylene (PP)

Polypropylene is a lightweight, flexible, and chemically resistant thermoplastic with excellent fatigue resistance, making it ideal for parts that need to bend repeatedly, such as living hinges. It is also highly resistant to moisture, acids, and solvents, and has good impact strength at room temperature. Common uses include food containers, caps and closures, automotive components, and medical syringes. Though not as stiff or strong as some other materials, its low cost and versatility make it one of the most widely used plastics.

Polyethylene (PE) – HDPE and LDPE

Polyethylene comes in several grades, with High-Density Polyethylene (HDPE) being more rigid and Low-Density Polyethylene (LDPE) offering more flexibility. Both types are chemically resistant, have low moisture absorption, and are easy to process. HDPE is commonly used in durable goods like fuel tanks and piping, while LDPE is suited for softer parts like squeeze bottles and packaging films. PE is widely used in consumer goods and industrial applications due to its low cost and chemical inertness.

Polystyrene (PS) – GPPS and HIPS

Polystyrene is a low-cost, easily processed thermoplastic available in two main forms: General Purpose PS (GPPS), which is rigid and transparent, and High Impact PS (HIPS), which is modified with rubber to improve toughness. GPPS is often used for clear containers, while HIPS is more suited for applications that require some impact strength, like appliance housings, display trays, and toys. While easy to mold and decorate, PS is brittle and has low resistance to heat and chemicals.

Polycarbonate (PC)

Polycarbonate is an extremely tough, transparent plastic with high impact strength and excellent heat resistance. It's often used as a lightweight, shatter-resistant alternative to glass. PC is suitable for optical applications like safety glasses and light lenses, as well as structural components in electronics and automotive interiors. It can withstand repeated impacts without cracking and has good dimensional stability, though it is prone to scratching and more expensive than other clear plastics.

Nylon (Polyamide – PA6, PA66)

Nylon is a strong, abrasion-resistant thermoplastic with excellent mechanical properties, especially for applications requiring durability and wear resistance. It performs well under mechanical stress, making it suitable for gears, bushings, and automotive parts. However, it tends to absorb moisture from the air, which can affect dimensional stability and mechanical properties. Nylon can be reinforced with glass fiber for added strength and stiffness in structural applications.

Acetal (POM – Polyoxymethylene)

Acetal, also known by trade names like Delrin®, is a stiff, low-friction plastic known for its excellent dimensional stability, chemical resistance, and wear properties. It's often used for precision components such as gears, bearings, fasteners, and valves. Acetal's low moisture absorption and high machinability make it ideal for applications requiring tight tolerances and mechanical strength. However, it is not recommended for prolonged exposure to strong acids or high temperatures.

PET (Polyethylene Terephthalate)

PET is a strong, lightweight, and transparent plastic with good moisture and gas barrier properties, commonly used in beverage bottles and food containers. In injection molding, PET can be used for durable consumer goods and packaging components that require clarity and strength. While it offers excellent chemical resistance and dimensional stability, it must be carefully dried before molding to avoid degradation.

TPU (Thermoplastic Polyurethane)

TPU is a highly flexible, abrasion-resistant material with rubber-like elasticity and excellent resistance to oils and greases. It combines the processability of thermoplastics with the performance of elastomers, making it ideal for overmolded grips, seals, gaskets, and protective cases. TPU can be either soft or rigid depending on formulation and is valued for its impact resistance and ability to return to its original shape after deformation.

PMMA (Polymethyl Methacrylate / Acrylic)

PMMA, often marketed as acrylic or by trade names like Plexiglas®, is a transparent plastic known for its optical clarity, UV resistance, and scratch-resistant surface. It is a cost-effective alternative to glass in applications like light lenses, displays, signage, and protective barriers. PMMA is more brittle than polycarbonate but offers excellent weatherability, making it a good choice for outdoor applications where clarity and appearance are important.

3.3. MATERIAL SELECTION GUIDE

When choosing a thermoplastic material, consider the following decision criteria:

Performance Attribute	Recommended Materials
High Impact Strength	PC, ABS, TPU
Chemical Resistance	PP, PE, POM
Dimensional Stability	ABS, PC, POM, PA66 (dry)
Optical Clarity	PC, PMMA, PET
Flexibility	PP, TPU, LDPE
High Heat Resistance	PC, PA66 (glass-filled), PPS (not listed above)
Fatigue Resistance (Hinges)	PP, Nylon, Acetal
Low Cost	PP, PE, PS
Cosmetic Finish / Paintable	ABS, PC, PMMA
Overmold Compatibility	TPU, TPE, soft durometer blends

For highly regulated applications (e.g., medical, food contact), always confirm certifications, biocompatibility, and regulatory approvals before finalizing material.

3.4. FILLERS AND REINFORCEMENTS

Fillers are additives introduced into thermoplastics to modify mechanical properties, thermal behavior, dimensional stability, and weight. The right filler improves performance and can reduce material cost, but also affects mold design and wear.

Glass Fillers

- Glass fibers increase tensile strength, stiffness, and dimensional stability while reducing creep.
- Common in Nylon, PBT, and PP grades, they can significantly improve heat deflection temperature (HDT).
- Downsides include increased tool wear and visible surface effects (fiber breakout, swirl marks).

Mineral Fillers

- Talc, calcium carbonate, and mica are used to improve rigidity, reduce shrinkage, and enhance surface finish.
- Often used in PP and PE compounds to increase dimensional control in structural applications.
- Minerals are less abrasive than glass but increase brittleness.

Carbon Fillers

- Carbon fibers provide high stiffness and electrical conductivity, often used in aerospace and high-performance parts.
- Carbon black improves UV resistance and imparts antistatic or conductive properties.
- Adds weight and cost, and may require specialized molding parameters.

Other Common Fillers:

- PTFE (Teflon): Improves wear resistance and reduces friction in gears and sliding parts.
- Stainless Steel or Bronze Powders: Used for EMI shielding or to increase thermal conductivity.
- Flame Retardant Additives: Required for compliance in electrical or regulated markets.
- Color Masterbatches: Pigments or dyes introduced for appearance, sometimes with functional UV inhibitors.

Note: Filled materials often increase viscosity, reduce flow length, and raise molding pressure. They also accelerate wear on mold surfaces, which must be countered with hardened tool steels (e.g., H13 or S7) and surface treatments.

3.5. SHRINK RATE OVERVIEW

Shrinkage is the reduction in part dimensions as the plastic cools and solidifies in the mold. It varies by material and part geometry and must be compensated for in mold design to maintain dimensional accuracy.

Material	Typical Shrink Rate (%)
ABS	0.4 – 0.7
Polypropylene (PP)	1.0 – 2.5
Polyethylene (HDPE)	1.5 – 3.0
Polystyrene (PS)	0.3 – 0.6
Polycarbonate (PC)	0.5 – 0.7
Nylon (PA6/PA66)	0.7 – 1.5 (dry); higher if wet
Acetal (POM)	1.5 – 2.1
PET	0.2 – 0.6
TPU	1.0 – 2.0
PMMA (Acrylic)	0.2 – 0.8

Shrinkage depends on part thickness, flow orientation, mold temperature, and fiber fill content (if reinforced). Always verify final dimensions using first-off-tool (FOT) measurements and adjust mold dimensions as needed.

4. DESIGN FEATURES

4.1. RIBS

Ribs are thin, wall-like features added to injection molded parts to increase structural rigidity without significantly increasing material usage or wall thickness. Their primary purpose is to strengthen large or flat surfaces that might otherwise be prone to bending, warping, or collapsing under load.

The key benefits of Ribs:

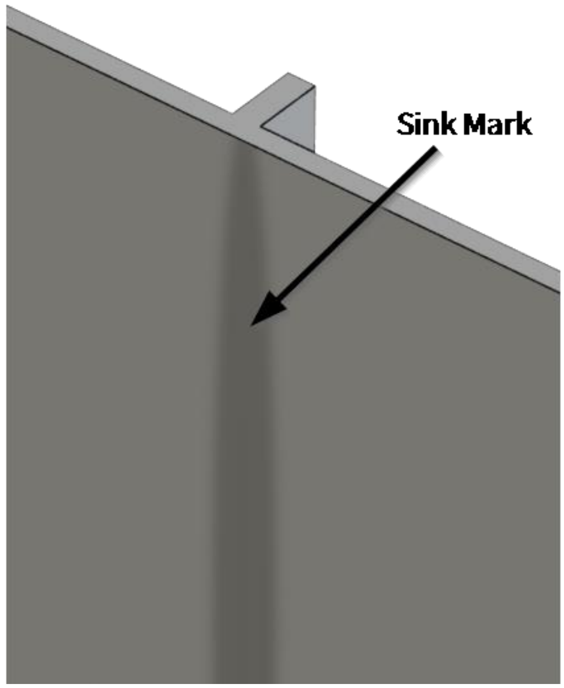
- Increase stiffness and strength of a part without adding mass.
- Support thin-walled features, preventing flex or deformation.
- Improve dimensional stability and reduce warping during cooling.

Ribs are a cost-effective way to reinforce parts while maintaining moldability and keeping material cost and weight low. Proper rib design is essential to balance mechanical performance with manufacturability.

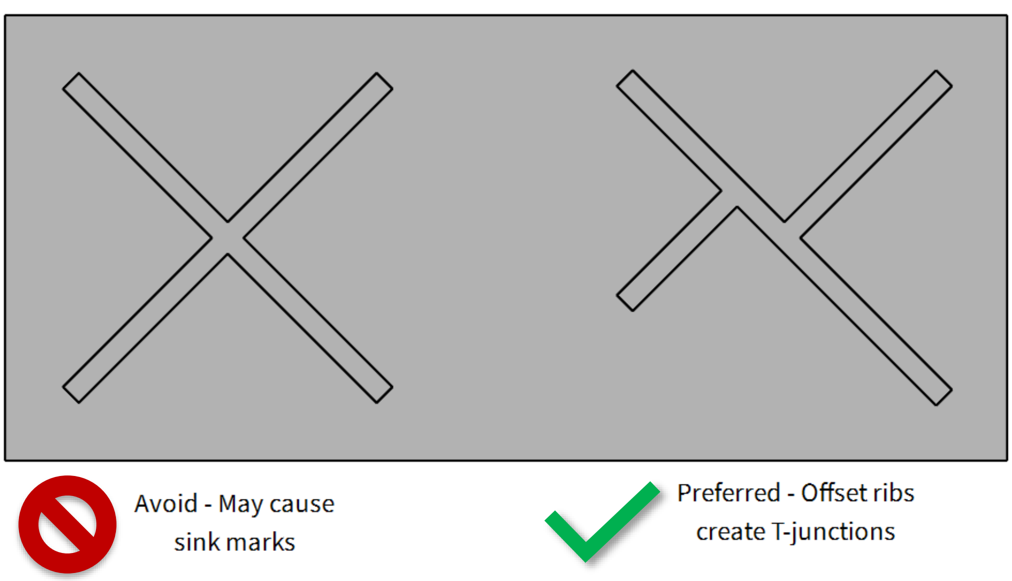
When designing ribs, the height of the rib should be less than 5 times the part thickness. A minimum draft angle of 0.5 degrees should be added to both sides of the rib, and the tip of the rib should not be thinner than 0.75mm or 0.03".

The thickness of the ribs should be between 50% and 75% of the part thickness. Ribs thinner than 50% may not fill properly, while ribs thicker than 75% can cause sink marks.

Sink marks are visual defects caused by a larger volume of plastic shrinking and pulling the material in while cooling down.



Ribs function by increasing the area moment of inertia to the part. Therefore, ribs intended to react to bending should be in line deflection of the part, while ribs intended to react to torsion should be designed in an angular pattern along the twisting axis.



4.2. BOSSES

Bosses are cylindrical features designed to receive fasteners such as threaded inserts or self-tapping screws. Threaded inserts are metal components embedded in plastic to provide durable threads for repeated fastening, while self-tapping screws cut their own threads directly into the plastic during installation.

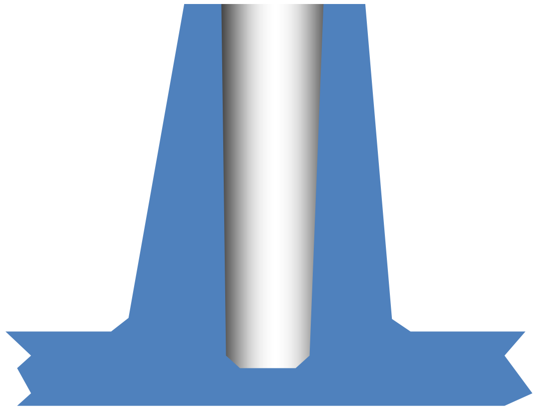
Bosses:

- Provide structural support for fasteners in plastic parts.
- Distribute load from tightening torque or pull-out force across the surrounding material.
- Enable disassembly and reassembly when used with threaded inserts.
- Serve as precise mounting points for mating components in an assembly.

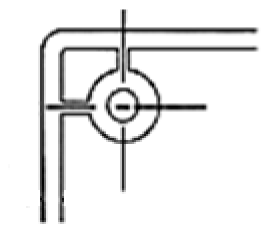
Self-tapping screws cut or form threads directly into the part during installation. Therefore, they can only be reused a limited number of times before the bosses become damaged. Three common types of self-tapping screws include Plastite, Hi-Lo, and PT threads.

Threaded inserts provide stronger and reusable threads to the plastic boss. They allow for repeated assembly without degrading the plastic and offer higher pull-out resistance compared to self-tapping screws. However threaded inserts must be installed using either a heat stake or ultrasonic staking machine. When threaded inserts are installed into bosses, molten plastic is forced down into the boss. Therefore, additional space is needed below the insert for the plastic to sit after installation.

When designing the boss, the wall thickness at the base of the boss should be between 50% and 75% of the part thickness. The draft angle should be at least 0.5 degrees on the exterior wall, and at least 0.25 degrees on the interior wall. The bottom of the interior hole should extend such that the wall thickness below the boss is 100% to 75% of the part thickness. The height of the boss should not exceed 5 times the part thickness.



Whenever possible, bosses should not be directly connected to external walls. Instead, bosses should be connected to the walls using ribs.



5. PART THICKNESS

5.1. DETERMINING PART THICKNESS

Uniform part thickness is one of the most critical principles in injection molded part design. Variations in part thickness create uneven cooling, which leads to sink marks, internal stresses, voids, and warpage. These issues not only compromise aesthetics and structural performance but also increase the likelihood of part rejection, tooling rework, and dimensional instability.

Wherever possible, maintain consistent thickness throughout the part. For most materials, a recommended thickness range is between 2.0 mm and 3.0 mm (0.080 in to 0.120 in). Specific materials may tolerate more variation, polypropylene can flow through thinner sections, while glass-filled nylons often require thicker sections for proper fill. When thickness transitions are unavoidable, they should be gradual, using a taper or fillet to prevent sudden changes in cooling rate.

Projected Surface Area (L × W)	Approx. Area	Min. Part Thickness
≤ 5 in × 5 in (127 mm × 127 mm)	25 in^2 (161 cm^2)	0.080 in (2.03 mm)
10 in × 10 in (254 mm × 254 mm)	100 in^2 (645 cm^2)	0.100 in (2.54 mm)
15 in × 15 in (381 mm × 381 mm)	225 in^2 (1452 cm^2)	0.120 in (3.05 mm)
> 225 in^2 (>1450 cm^2)	Large parts	Supplier dependent

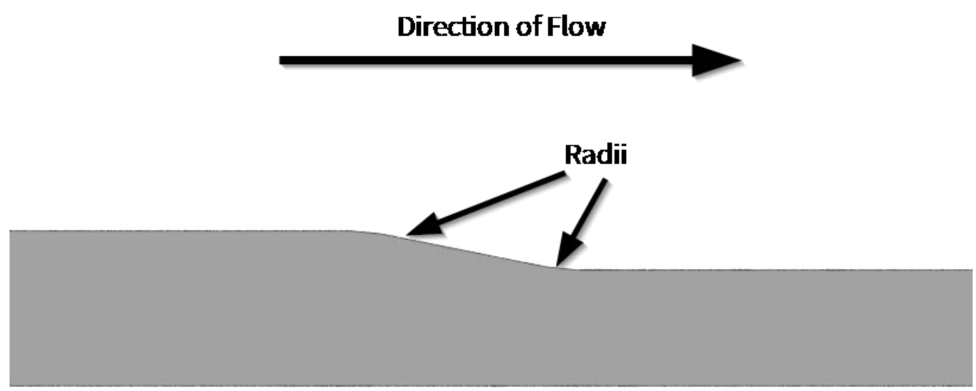
Thicker sections do not necessarily result in stronger parts. Instead, they slow cooling, increase cycle time, and create areas susceptible to sink. It is often more effective to use ribs and gussets to reinforce strength without increasing part thickness. These features improve stiffness while maintaining flow efficiency and minimizing shrinkage defects.

Ultimately, part thickness is not just a mechanical consideration; it also drives material choice, tooling complexity, and cycle efficiency. Keeping it consistent and controlled is one of the most reliable ways to reduce risk, improve quality, and support high-yield production.

5.2. NON-UNIFORM PART THICKNESS

When different part thicknesses are required, the change in thickness should happen through a gradual transition. The gradual transition helps prevent turbulence and back pressure as the mold is filled, which can cause warpage and stress cracks after the part is formed.

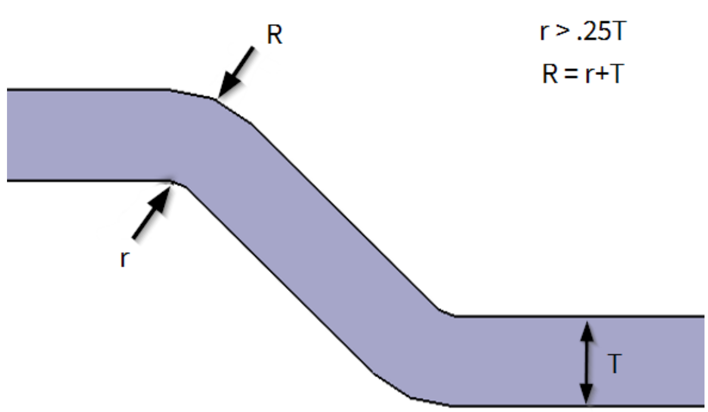
- A slope should be designed between the two thicknesses with the length of the slope being at least 5 times the change in thickness.
- Ideally, the thinner section should not be thinner than 85% of the thicker section.
- As the mold fills with plastic, the plastic should flow from thick to thin section. Meaning the mold gates should be located at the thickest section of the part.
- Corner radii should be added to the corners of the transition to reduce flow turbulence.



5.3. CORNER TRANSITIONS

As the part wall changes direction, the wall thickness should remain constant. Sharp external corners cause a non-constant wall thickness while sharp internal corners cause flow turbulence and back pressure. Both the internal and external corners should have gradual radii.

- The internal radius should be greater than 25% of the wall thickness
- The external radius should be equal to the internal radius plus the wall thickness

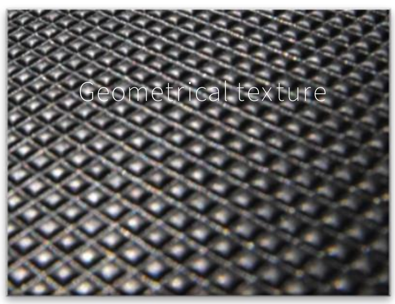


6. TEXTURE

Texturing is a surface finish applied directly to the mold cavity, typically using acid etching or laser engraving, to create visual and tactile characteristics on the molded part. It serves both functional and aesthetic purposes.

Texturing can:

- Improve the part's appearance by emulating materials like leather, wood, or stone
- Enhance grip and tactile feel
- Conceal cosmetic defects such as sink marks, parting lines, or flow lines
- Increase friction between mating surfaces
- Minimize the need for post-processing in adhesive bonding



When applying a texture, it is important to remember that deeper textures may require steeper draft angles.

Mold-Tech Reference	Average depth (microns)	Minimum suggested draft angle
MT 11000	10	1.5°
MT 11010	25	2°
MT 11020	40	3.5°

Note: When applying texture to the core side of the mold, the recommended draft angle should be doubled to ensure proper part ejection.

Design and Manufacturing Best Practices

- Early specification: Texture selection should be finalized before mold cutting. Late changes may require re-polishing and re-etching at significant cost.
- Avoid abrupt transitions: Sharp changes between textured and polished areas can create visible parting lines or flash traps. Use radii or step features to blend transitions smoothly.
- Material behavior: Some plastics replicate texture with higher fidelity than others. For example, rigid amorphous materials like ABS and PC hold detail better than softer elastomers.
- Text above functional areas: Avoid applying texture on sealing surfaces, wear points, or areas requiring dimensional precision unless validated through testing.

Specifying Texture in CAD and Drawings

Industry standards such as Mold-Tech, VDI 3400, and SPI should be used when specifying surface finishes. These can be applied:

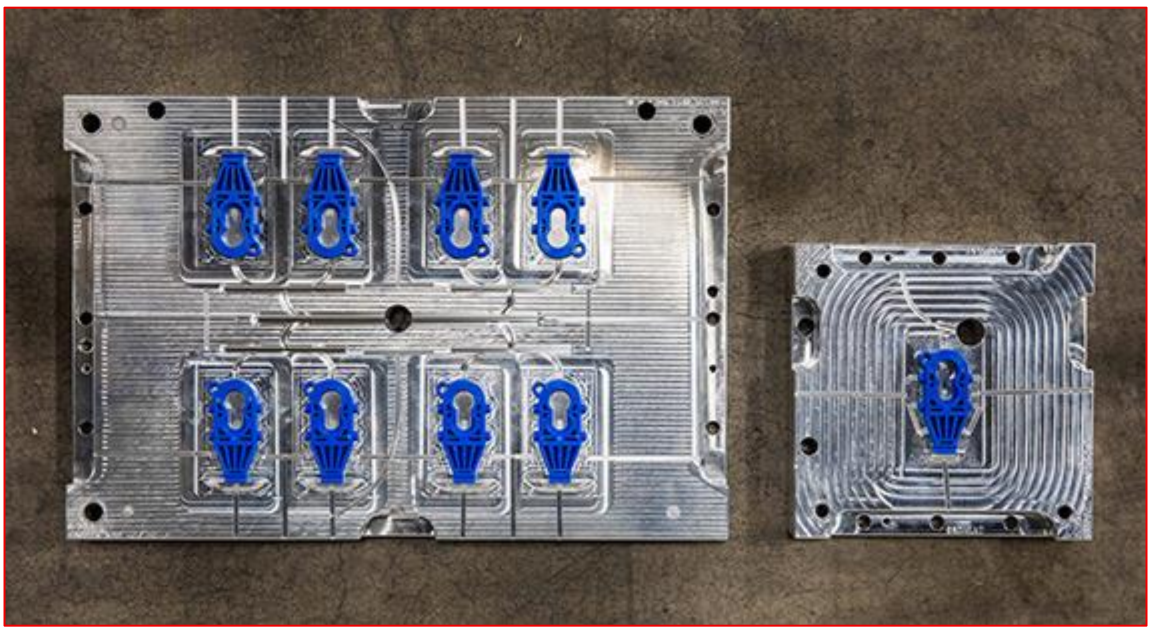
- As texture callouts in 2D technical drawings
- As mapped areas or annotations in 3D CAD models
- Along with defined draft angles and parting line strategies in the DFM stage

7. COST DRIVERS

Injection molding offers low per-part costs at high volumes, but the upfront investment in tooling and the long-term cost of production can vary significantly based on design choices. Understanding these cost drivers allows design teams to optimize for performance without inflating the budget.

The complexity of the mold is one of the most significant cost factors. A simple single-cavity mold with no side actions is far less expensive than a multi-cavity mold with slides, lifters, inserts, or internal cooling lines. Every moving component increases machining time, mold maintenance, and risk of failure over time. Whenever possible, designs should aim to avoid undercuts or internal features that require complex tooling. Strategically using ribs, bosses, and core pulls can reduce the need for side actions while achieving the same functional result.

- Multi-cavity molds reduce part cost but significantly increase tooling investment.
- Features like slides, lifters, inserts, and textured finishes add to tool complexity and cost.
- Design simplifications that avoid undercuts or eliminate side actions can drastically reduce tool build time, maintenance, and overall cost.



Part geometry also plays a central role in cost. Thin-walled parts require precise control over mold temperature and injection pressure to ensure proper fill, increasing cycle time and tooling wear. Parts with inconsistent wall thickness may shrink unevenly, leading to warpage or sink marks, and requiring mold rework or redesign. Tight tolerances can dramatically increase tooling costs as higher precision machining and more rigorous quality checks are needed. Where possible, tolerances should be relaxed unless they are functionally critical.

- Thin-walled parts require more precise temperature and pressure control, increasing cycle time.
- Uneven wall thickness leads to defects like warpage and sink, requiring mold rework.
- Overly tight tolerances add cost in both tool precision and inspection requirements.

Surface finish and texture requirements directly influence the mold manufacturing process. High-gloss or optical surfaces demand high-grade tool steel and extensive polishing, while deep textures (such as Mold-Tech patterns) require etching and additional draft angle to ensure proper release. These requirements increase both mold fabrication time and complexity.

- High-gloss or textured finishes increase mold polishing time and may require specific steel grades.
- Deeper textures require greater draft angles and influence mold design.

Material choice can affect both tooling and part cost. Some materials, like glass-filled nylons or flame-retardant ABS, are abrasive or corrosive to molds, accelerating tool wear and requiring hardened steel like H13 or coatings like nitriding. In contrast, commodity plastics like polypropylene or polyethylene can run in lower-cost aluminum tools for prototyping or short runs. Material shrinkage rates also vary, affecting tool size and tolerance allowances, which can lead to added design iterations if not accounted for.

- Material cost varies widely, and specialty or reinforced plastics may require hardened molds or longer cycle times.
- Filled or abrasive materials cause faster tool wear, requiring tool steels like H13 or P20 with surface treatments.

Finally, cycle time and volume play a key role in amortizing the tool cost. High-cavitation molds and hot runner systems reduce per-part cost by maximizing output, but only make economic sense for production volumes above 100,000 units. For lower volumes, bridge tooling, aluminum molds, or even 3D printed inserts can be used to reduce tooling investment while validating the design.

- High-volume production justifies complex, high-cost tools due to amortization.

- Shorter cycle times reduce per-part cost but require optimized mold cooling and ejection.

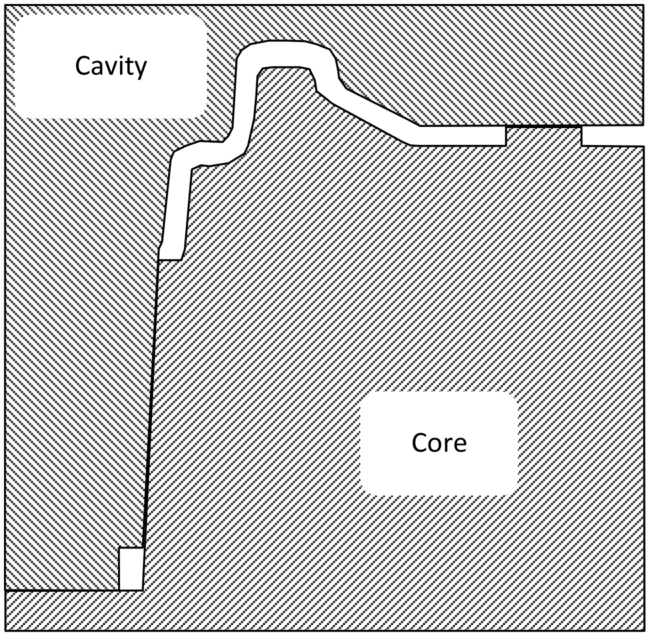
8. UNDERCUTS AND HOLES

Undercuts are features on a molded part that prevent it from being ejected straight out of the mold along the tool's primary opening direction. These include internal or external projections, hooks, holes, snap-fits, or grooves that interfere with the mold's ability to separate cleanly. While undercuts are often necessary for part function, such as for locking features, assembly retention, or aesthetic detailing, they introduce additional complexity to both part design and mold construction.

To mold undercuts, mechanical actions must be used to physically move components of the tool away from the part before ejection. Common solutions include slides, which move perpendicular to the tool direction to form side holes or recesses, and lifters, which move at an angle to release internal catches or angled surfaces. Both add cost, extend mold lead time, and increase the chance of maintenance or alignment issues. When planning an undercut, it's important to evaluate whether the feature can be reoriented, replaced with a simpler alternative (such as a detent or overhang), or eliminated entirely.

Undercuts should be treated as intentional, engineered decisions, not as incidental byproducts of complex geometry. Thoughtful planning can reduce tooling complexity, improve mold durability, and ensure parts are manufacturable at scale.

Holes in sidewalls can be formed using direct contact between the cavity and the core. When designing features where the core and cavity are in direct contact, there must be a minimum draft angle of 3 degrees. This 3-degree angle helps reduce friction between the mold surfaces, preventing excessive wear on the mold.



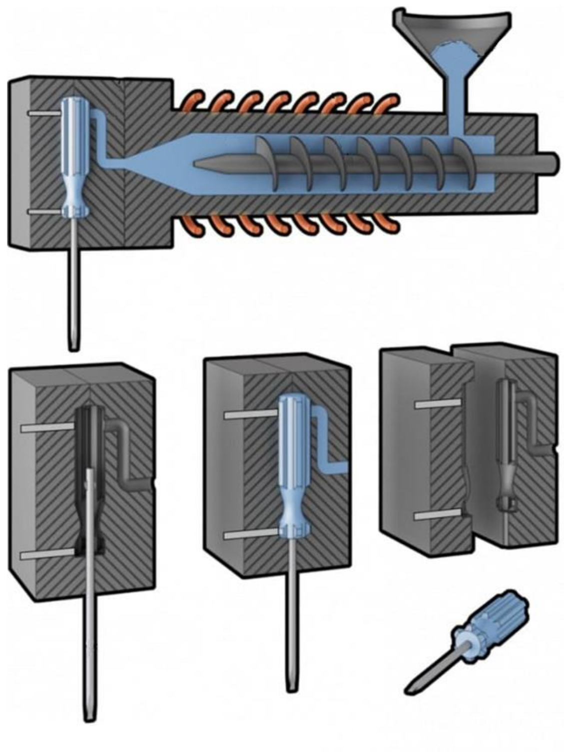
9. OVERMOLDING

Overmolding is the process of molding one material over another in a multi-shot or insert-based sequence to create a unified component with enhanced functionality, aesthetics, or ergonomics. It is commonly used to combine rigid structural substrates with soft-touch grips, seals, or color accents. Overmolding can also bond dissimilar materials to eliminate fasteners and simplify assembly.

Successful overmolding requires careful material selection to ensure chemical compatibility and adhesion between the base and overmolded materials. Thermoplastic elastomers (TPE or TPU) are frequently used in soft-over-hard applications, while engineered plastics like PC, ABS, or Nylon serve as common base substrates. Poor bonding, warpage, or delamination can occur if thermal, mechanical, or surface preparation parameters are not well-controlled.

From a tooling perspective, overmolding can be executed in two ways: multi-shot molding within a single tool, or insert overmolding, where a pre-molded part is manually or robotically loaded into a second cavity. Insert overmolding is more flexible for low-to-mid volume production, while multi-shot tools are best suited for high-volume, automated runs.

Overmolding improves perceived quality, enables design differentiation, and reduces assembly steps. However, it introduces process complexity and extended cycle time, making it essential to weigh functional benefits against cost and manufacturing efficiency during the early design phase.



10. TOLERANCING

Injection molding offers repeatable precision for high-volume production, but it does not provide the same inherent dimensional control as machining. Tolerances are influenced by material shrinkage, mold wear, cooling, and part geometry. Setting appropriate tolerances is critical to avoiding over-engineering, which can increase tooling costs and inspection requirements.

For most thermoplastics, standard achievable tolerances for molded parts are as follows:

Feature Type	Typical Tolerance Range (mm)	Typical Tolerance Range (inches)
Linear dimensions (≤100 mm, 4 in)	±0.05 mm to ±0.20 mm	±0.002 in to ±0.008 in
Hole diameter (as-molded)	±0.075 mm to ±0.25 mm	±0.003 in to ±0.010 in
Wall thickness	±5% to ±20% of nominal value	±5% to ±20% of nominal value
Flatness / perpendicularity	0.1 mm to 0.5 mm	0.004 in to 0.020 in
Boss and snap-fit features	±0.05 mm to ±0.10 mm	±0.002 in to ±0.004 in (critical features)

Tighter tolerances may be achievable with stable materials (e.g., ABS, POM), controlled environments, and post-mold machining. For functional features, always confirm tolerance requirements with your mold maker to balance precision with cost and mold life.

Factors Affecting Tolerance

- Material shrinkage varies from 0.2% to 3.0% depending on polymer type, filler content, and flow orientation.
- Mold temperature and cooling channel layout impact part consistency and warpage.
- Part geometry plays a role, thin or unsupported sections are more prone to deflection during ejection and cooling.
- Mold precision and maintenance affect repeatability over the tool's lifespan, especially in high-cavitation molds.

Critical vs. Non-Critical Dimensions

Use general tolerances for non-functional dimensions and tight tolerances only where required, such as bearing seats, press fits, or sealing features. Coordinate with your tooling partner during the DFM review to confirm which dimensions require steel-safe conditions or post-processing.

Post-Mold Operations for Tight Tolerances

When tolerances beyond molding capabilities are necessary, consider secondary machining (e.g., boring, reaming, trimming) or insert molding, which allows precise features to be molded around a pre-machined metal or plastic insert.

Best practice: Define tolerances based on functional need and molding capability, not arbitrary default values from CAD. Where precision is critical, document the requirement as “critical-to-function” (CTF) and communicate it clearly in drawings and technical reviews.

11. ASSEMBLY AND JOINING METHODS

Injection molded components are often part of larger assemblies, and their design should account for how parts will be joined, fastened, or bonded during the product’s lifecycle. Understanding joining methods early in the design process helps reduce part count, improve manufacturability, and eliminate unnecessary post-processing steps.

One of the most efficient and cost-effective methods of assembly in molded parts is the snap-fit. Snap-fits use elastic deformation to temporarily displace one feature during assembly, which then returns to its original shape to lock the parts together. The most common type is the cantilever snap-fit, where a flexible beam locks into a mating groove. This design is ideal for access panels, battery doors, or housings requiring frequent assembly and disassembly. Annular snap-fits are used in circular parts such as caps or cylindrical containers, where the male and female parts snap together via circumferential features. For rotating or hinged applications, torsional snap-fits use a twisting motion to engage the latch. While snap-fits reduce hardware and simplify assembly, they must be carefully analyzed for stress, fatigue, and retention force over repeated cycles.

11.1. SNAP-FITS

Snap fits are commonly used in injection-molded parts to provide quick, tool-free assembly while eliminating the need for fasteners or adhesives. When designed properly, snap features offer a reliable, repeatable locking mechanism with excellent assembly efficiency and cost savings. However, poorly designed snaps can lead to part damage, inconsistent engagement, or tooling challenges.

Design tips:

- The width of the snap arm should be at least one-third the height to ensure structural stability and reduce stress concentration during deflection.
- Draft Angles:
  - Metal-to-Plastic Surfaces (e.g., inner faces): Apply at least 0.5° draft to allow clean ejection from the mold.
  - Metal-to-Metal Surfaces (e.g., when cavity and core touch): Require a minimum 3° draft to prevent sticking or tool wear. This is especially critical on deep features and snap latches where ejection resistance is higher.
- The snap hook creates an undercut that must be designed to allow smooth deflection and engagement without exceeding the material's elastic limit. Consider using fillets at the base of the snap to reduce stress risers and improve fatigue life.
- Use flexible, fatigue-resistant plastics such as polypropylene (PP), acetal (POM), or nylon (PA) for snaps. Avoid brittle materials that may crack under repeated flexing.
- Ensure that undercuts created by snap hooks are accessible with straight-pull tools, or use slides/lifters if the snap geometry prevents direct release. Avoid overhanging hooks or sharp corners that complicate mold ejection.
- Snap features should be designed to deflect within the elastic range of the material, with an acceptable assembly force (typically under 30–40 N for handheld components). Over-stressing can lead to creep or permanent deformation.

11.2. PRESS FITS

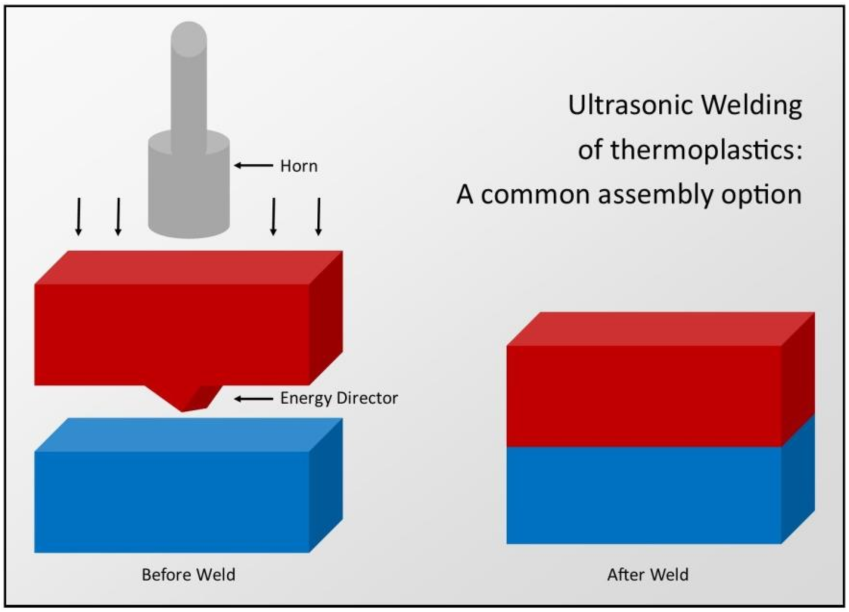
Press-fit features rely on interference between mating parts. Dimensional control and material selection are critical to avoid stress cracking or misalignment. They are common in hub-shaft or bushing assemblies.

11.3. THREADED FASTENERS

Bosses can be designed for self-tapping screws or heat-set threaded inserts, enabling removable or serviceable assemblies. Reinforce critical bosses with ribs and avoid excessive torque.

11.4. WELDED JOINTS

- Ultrasonic Welding: Uses high-frequency vibrations to bond parts at the molecular level; suitable for ABS, PC, PP, and other thermoplastics.
- Heat Staking: A heated tool softens a plastic post to permanently join it with another component.
- Adhesives & Solvent Bonding: Useful for dissimilar materials or when mechanical methods aren't feasible; requires attention to surface prep and alignment.



12. ADDITIONAL TOOLING CONCEPTS

12.1. INJECTION GATE

Injection gates control the flow of molten plastic as it enters the mold cavity. They should be strategically placed in the thickest section of the part to promote even flow and reduce the risk of voids or sink marks. Once removed, gates leave a visible gate scar. Therefore, gates should not be located on visible surfaces or mating surfaces where surface finish is critical.

Additionally, gates should be located to minimize weld lines and allow trapped air to escape through venting, avoiding surface defects and incomplete fills.

The most common type of gate is the tab gate, or edge gate. An edge gate is placed on the outer edge of the part and allows plastic to flow directly into the cavity. It's simple to design and machine, making it a common choice for most parts.

Other types of gates include Submarine, fan, direct sprue, pin and diaphragm gates.

Submarine Gate (Tunnel or Banana Gate) – Submarine gates enter the part from below the parting line at an angle and automatically shear off during ejection. They are ideal for high-volume production where automatic degating is preferred. While they leave minimal visible marks, they are more complex to machine and can be fragile in brittle or filled materials.

Fan Gate – A fan gate is a widened, shallow gate that distributes flow evenly across a broad edge. It is especially useful for thin-walled or wide parts that are prone to warping. Though it improves flow distribution, it typically leaves a larger gate vestige and requires trimming.

Direct Sprue Gate – The direct sprue gate connects the sprue directly to the part, commonly used in single-cavity molds. It offers low resistance and efficient filling for large parts but leaves a noticeable mark and may require post-processing to remove the gate vestige.

Pin Gate (Hot Tip or Cold Pin) – Pin gates are used in hot runner systems and deliver plastic to the center of a part, often resulting in a clean, centrally located gate mark. They are ideal for round parts and allow balanced filling across multiple cavities. Precise thermal control is essential to prevent stringing or drool at the gate.

Diaphragm Gate – Diaphragm gates form a circular opening around a core and are used for cylindrical or hollow parts. They ensure uniform filling around the entire perimeter, which helps avoid warping or uneven shrinkage. These gates can be difficult to remove cleanly and often leave a ring-shaped vestige.

12.2. MOLD INSERTS

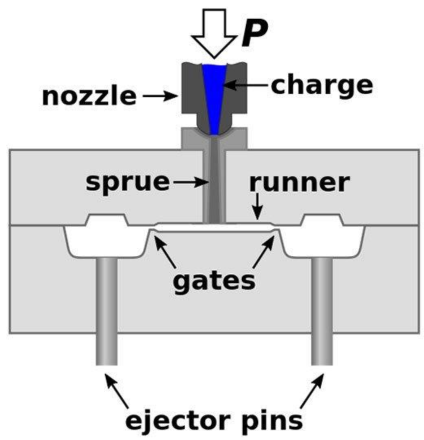
Mold inserts are removable metal inserts installed into the cavity or core side of a mold to form localized features. They provide flexibility in tool design, allowing specific areas to be swapped, modified, or upgraded without machining a new tool. Inserts are particularly useful in high-wear areas, fine detail zones, or regions likely to change during production—such as logos, date stamps, or part revisions.

By isolating risk-prone geometry, mold inserts reduce rework cost and shorten iteration timelines. For example, if a hole diameter or clip geometry needs to be adjusted after initial sampling, only the insert needs to be replaced rather than re-cutting a full cavity block. This modularity is valuable in both development-stage tools and long-life production molds with changeable product configurations.

Inserts also allow for targeted material upgrades in specific regions. If a part contains deep, narrow features or thin-walled bosses that experience high ejection forces, the insert can be manufactured from hardened steel or coated for increased durability without upgrading the entire mold. They can also improve mold maintenance by enabling local polishing or repairs without full tool disassembly.

12.3. EJECTORS

Ejector pins are mechanical components used to push the molded part out of the mold cavity once it has cooled. Located on the core side of the mold, they activate as the mold opens and the ejector plate moves forward, gently forcing the part off the core features. Ejector pins are essential for automated part removal and help maintain high production efficiency. They must be placed carefully to avoid visible marks on critical surfaces and to apply even pressure that prevents part distortion or damage during ejection. Proper ejector pin design ensures smooth demolding and protects both the part and the mold from unnecessary wear.



Ejector pads are flat, broad-surfaced components used to push molded parts out of the mold cavity during the ejection phase of the molding cycle. Unlike ejector pins, which concentrate force on small points, ejector pads distribute the ejection force over a larger area, making them ideal for removing parts with flat or thin surfaces that are prone to deformation, scuffing, or breakage. They are commonly used on parts with large planar faces or cosmetic surfaces where pin marks would be unacceptable. Ejector pads are typically integrated into the mold's ejector plate system and are often custom-machined to match the contour of the part, ensuring smooth, uniform release without compromising surface quality.

13. PRODUCTION PLANNING

Effective production planning is essential to minimize delays, reduce tooling risks, and ensure on-time product launches. From initial concept to final delivery, understanding the typical lead times and milestones associated with injection molding enables better scheduling and resource allocation.

The first consideration is tooling lead time. Prototype aluminum tools, often used for low-volume runs or functional validation, can be machined in as little as 2–4 weeks. However, production-grade steel tools, especially those with multiple cavities, textured surfaces, or side actions, typically take 6–12 weeks to fabricate. These timelines include design-for-manufacturability (DFM) reviews, mold design, CNC machining, polishing, and initial T1 sampling. Delays in any stage, such as material availability, design changes, or approvals, can impact the entire production schedule.

- Prototype aluminum tools: 2–4 weeks (depending on complexity and vendor availability)
- Production steel tools: 6–12 weeks (including DFM, cutting, and T1 trials)
- Complex multi-cavity or tool-steel hardened molds may take longer and require more validation loops.

Order volume also affects tool design and cost strategy. For limited runs (<1,000 parts), soft tooling or insert-based modular molds are often the most economical. For mid-volume production (10,000–100,000 units), a single-cavity or dual-cavity mold made from P20 steel offers a balance of durability and cost. For high-volume production, tools are typically built from hardened steel (H13 or S7), with hot runners, cooling channels, and automated ejection to support cycle times under 30 seconds. These tools may cost more initially, but their longer lifespan and higher throughput reduce long-term cost per unit.

- Low-volume production (<1,000 units) is best served with soft tooling or bridge tools.
- Mid-volume (10,000–100,000 units) often uses P20 steel with moderate automation.
- High-volume (>100,000 units annually) justifies multi-cavity, hot-runner systems, and hardened steel (H13 or S7) molds.

Design teams should account for revision planning when finalizing CAD models. Late-stage changes, especially those involving boss locations, mounting features, or mating geometries, can require mold rework or complete tool replacement. For this reason, mold designers often use modular inserts for areas likely to change, such as logos, text, or fastener locations. This allows simple steel swaps rather than full cavity rebuilds. Engaging in early DFM review with your mold maker or engineering consultant greatly reduces risk of costly revisions later.

- Design changes post-tool-cutting can be costly and time-consuming.

- Plan for modular inserts in areas likely to change (logos, mounting features, alignment pins).
- Review DFM early with mold maker or consult an experienced design firm to prevent costly tooling revisions.

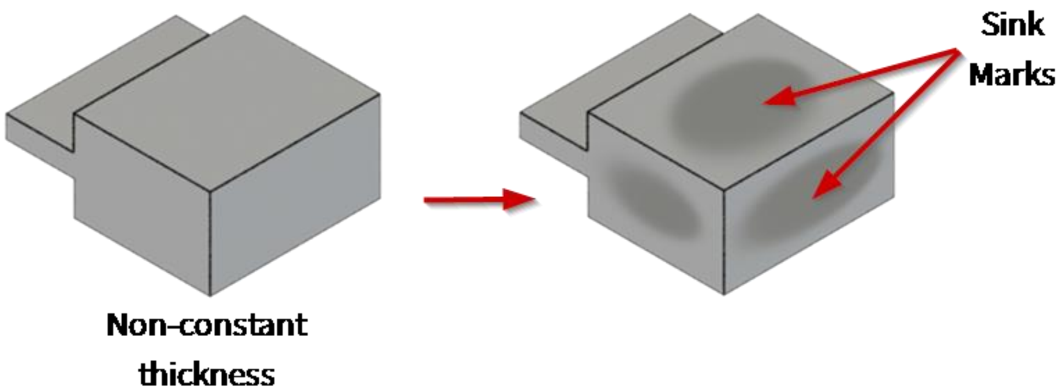
Once the mold is complete, first-off-tool (FOT) samples are produced to validate form, fit, and function. Depending on part complexity, this may be followed by a series of optimization runs before moving to production. In regulated industries (e.g., medical, aerospace), formal qualification protocols like IQ/OQ/PQ or PPAP must be planned and executed, adding time and validation cost to the project.

- Plan for first-off-tool (FOT) samples for dimensional checks, fit testing, and cosmetic review.
- Allow 1–2 weeks for minor mold adjustments between FOT and production release.
- Regulated products (e.g., medical or aerospace) may require formal IQ/OQ/PQ or PPAP protocols.

14. COMMON DEFECTS

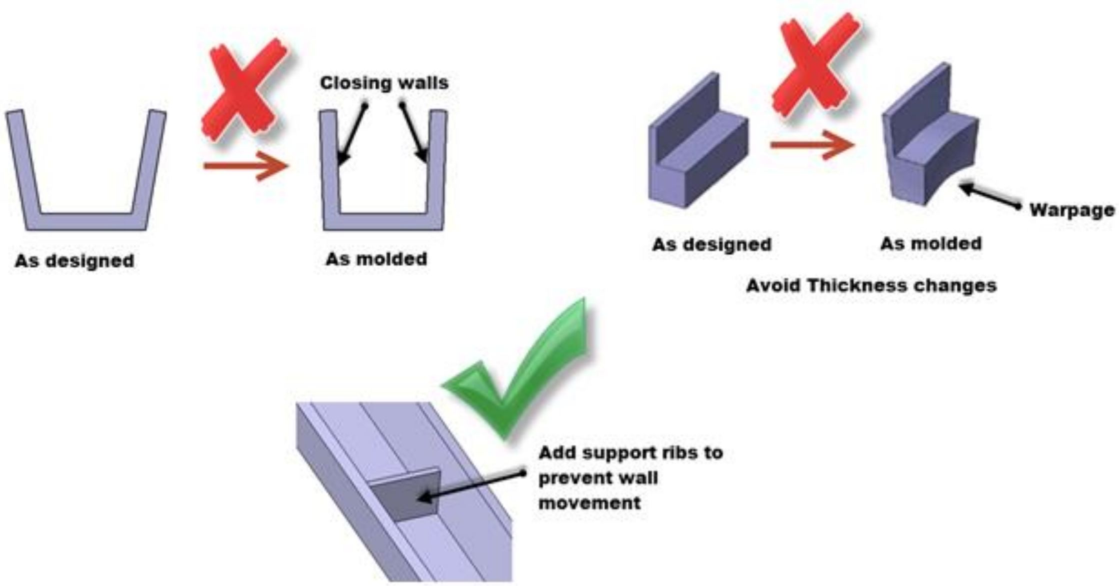
14.1. SINK MARKS

As previously described, sink marks are small depressions or dimples that appear on the surface of an molded part, typically above thick or heavily ribbed areas. They occur when the inner material cools and shrinks more than the outer surface, causing the surface to cave in slightly as the internal volume contracts. Sink marks are most common in areas where the wall thickness is inconsistent or where features like ribs, bosses, or thick sections create localized hotspots. These cosmetic defects can also affect part strength and fit, making it important to design with uniform wall thickness, use proper rib-to-wall ratios, and optimize gate placement and cooling to minimize their occurrence.



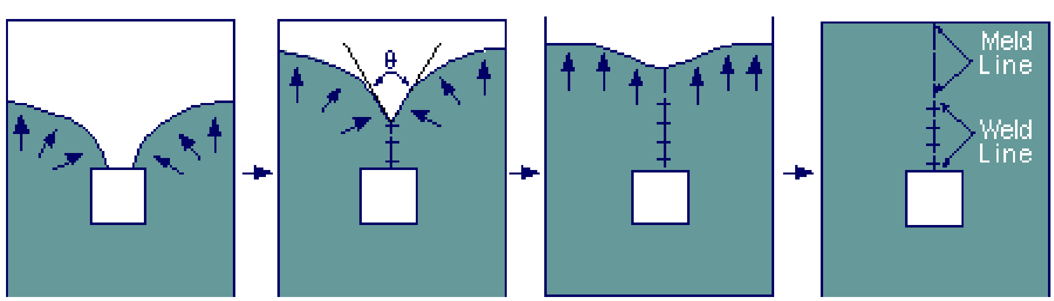
14.2. WARPAGE

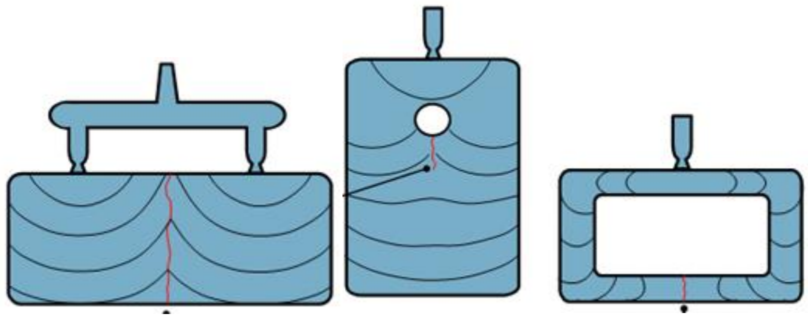
Warpage is the distortion or twisting of an injection molded part that occurs when different areas of the part cool and shrink unevenly. This uneven shrinkage creates internal stresses that cause the part to bend, curl, or deviate from its intended shape. Warpage is often caused by non-uniform wall thickness, poor cooling channel design, improper gate location, or the use of materials with high shrinkage rates. It can lead to dimensional inaccuracies, poor fit with mating components, and even functional failures. To reduce warpage, designers should aim for uniform wall sections, balanced mold filling, and controlled cooling throughout the part.



14.3. WELD LINES

Weld lines, also called knit lines or flow lines, occur when two or more flow fronts of molten plastic meet and solidify without fully bonding, which may result in a visible line or structural weak spot on the surface of an injection molded part. They typically form around holes or other internal features that divide the flow of material during mold filling. Weld lines can compromise both the appearance and structural integrity of a part. To minimize weld lines, designers and mold makers can adjust gate locations, increase melt temperature or injection speed, and ensure proper venting so the material flows meet while still hot and pressurized enough to fuse properly.





14.4. SPLAY

Splay is a cosmetic defect that appears as silvery streaks, lines, or cloudy marks on the surface of an injection molded part. It typically occurs when moisture, air, or volatile contaminants are present in the melt stream and become trapped in the resin during injection. As the plastic flows into the mold, these gases are rapidly compressed and heated, forming bubbles that streak along the surface and solidify in the flow direction.

Splay is most commonly caused by resin moisture, especially in hygroscopic materials like nylon (PA), PET, or PC, which absorb ambient moisture over time. If the material is not properly dried before molding, the moisture turns to steam under heat and pressure, resulting in surface blemishes. Other causes include excessive shear (due to high screw speeds or nozzle pressure), material degradation from overheating, or contamination from regrind and incompatible resins.

To avoid splay:

- Orient gates and flow paths away from cosmetic surfaces. Position the gate so that high-visibility areas are filled last, reducing the likelihood that splay streaks appear where they matter most.
- Avoid abrupt changes in wall thickness. Use gradual transitions and radii between thick and thin areas to reduce shear and turbulence that can trap gas or moisture.
- Enable proper mold venting through thoughtful geometry. Design parting lines and features that give mold makers options to place vents in non-critical locations for gas escape.

- Maintain adequate wall thickness near the gate. Thin sections at or near the gate increase fill pressure and shear. Thicker walls help reduce turbulence and promote even flow.
- Apply texture or matte finishes on visible surfaces. Mold textures can help mask minor cosmetic defects like splay, especially on external housing or enclosure parts.
- Limit long, thin flow paths when possible. Extended flow through narrow sections increases shear and moisture separation. Break up long flow paths with ribs or alternate gate locations.

While splay does not typically affect part strength or function, it can severely impact cosmetic appearance, especially on high-visibility or consumer-facing surfaces. Preventing splay is often a matter of disciplined material handling, stable processing conditions, and regular maintenance of molding equipment.

14.5. SHORT FILL

A short fill occurs when molten plastic fails to completely fill the mold cavity, resulting in an incomplete part. This defect typically appears as missing material at the far ends of the flow path, thin edges, or complex features. Short fills are a critical failure mode, as they render the part unusable and often indicate a breakdown in material flow, pressure, or moldability.

Short fills can be caused by several factors, including:

- Insufficient injection pressure or speed
- Material cooling too quickly before full cavity fill
- Inadequate gate or runner sizing
- Improper venting, causing trapped air to resist flow
- Excessive flow length relative to wall thickness
- Low melt temperature or degraded material viscosity

While processing adjustments can often correct short fill issues, part design plays a key role in preventing them before they occur.

Design Strategies to Prevent Short Fill:

- Maintain uniform wall thickness. Inconsistent walls can cause localized cooling and pressure loss. Uniform walls allow steady flow and reduce the chance of early freeze-off.
- Avoid overly thin sections. Thin walls require higher pressures and faster fill speeds. Keep wall thickness within material-specific guidelines (typically 2.0–3.0 mm) and avoid unnecessary thinning in long-flow areas.
- Use generous radii and flow-friendly transitions. Sharp corners or abrupt changes in geometry disrupt flow. Use fillets and gradual transitions to maintain material momentum.
- Shorten flow paths where possible. Break up long, flat surfaces with ribs or flow leaders, or use multiple gates to reduce overall flow distance and improve fill balance.
- Enlarge gates and runners as needed. Undersized gates or runners restrict flow and reduce packing pressure. Proper gate sizing ensures full cavity fill and minimizes flow hesitation.
- Ensure proper venting in difficult-to-fill areas. Trapped air can prevent plastic from filling thin or remote features. Design the part to allow for strategic vent placement by the mold maker.

Short fill defects are both a processing and design concern. By considering flow dynamics during the early design phase and collaborating with your tooling partner, you can significantly reduce the risk of short shots and ensure a robust, repeatable molding process.

15. CONCLUSION

Injection molding remains one of the most effective and scalable manufacturing processes for plastic components—but it is also one of the least forgiving. Every decision, from wall thickness and draft angles to gate placement and material selection, has an impact on part quality, tooling complexity, production cost, and overall product success.

This guideline has outlined the key principles of injection mold design, from Design for Manufacturability (DFM) to defect prevention, tolerance planning, material behavior, and assembly considerations. It has also highlighted how thoughtful part design—supported by a clear understanding of the molding process—can eliminate avoidable revisions, reduce development cycles, and deliver production-ready solutions.

Whether you're developing a single prototype or preparing for high-volume production, making informed design decisions early pays exponential returns. Great parts aren't just designed—they're engineered with intent, balance, and an understanding of real-world manufacturing constraints.

At I.M. Bohannon Mechanical Design, we partner with engineers, product teams, and founders to turn concepts into manufacturable components. With a deep background in mechanical design, prototyping, and production support, we help clients:

- Evaluate and optimize part geometry for injection molding
- Design tooling-ready CAD with appropriate draft, wall structure, and tolerances
- Select materials based on performance, cost, and shrink behavior
- Communicate effectively with toolmakers and suppliers
- Develop scalable, production-friendly product assemblies

Whether you're starting with a sketch or finalizing a drawing package, we can help close the gap between design and manufacturing—bringing products to market with speed, confidence, and precision.

Need help with your next injection molded product?

We offer:

- DFM reviews for molded parts
- Tooling-ready CAD and drawing packages
- Material selection and shrinkage analysis
- Supplier consultation and production planning
- Full-cycle product development support