

Injection Molding Design Guide

Design considerations for rapid manufacturing
of plastic parts using injection molding



Table of contents

1	Injection mold tooling process comparison	3
2	Size considerations	4
3	Considerations for undercuts	5
4	Other generic considerations	6
	Parting line limitations	6
	Deep ribs	6
	Rounded corners	6
	Sharp corners	7
	Rib-to-wall thickness ratios	7
	Warp	7
	Sufficient draft	7
5	Gating	8
6	Resin selection	8
7	Surface finish section	10
	Texturing	10
	Polishing	10
8	Lead time	11

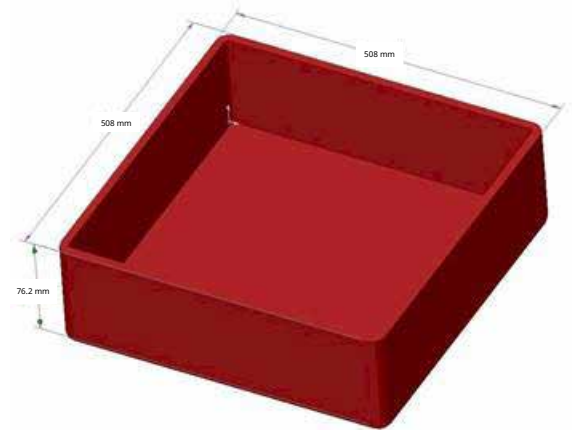
1 Injection mold tooling process comparison

	Rapid Injection Molding	Low-Volume Injection Molding	Production Injection Molding
Part size limitation	508 x 508 x 76.2 mm	914.4 x 914.4 x 381 mm	914.4 x 914.4 x 381 mm
Part geometry limitation	No undercuts causing slide action in tooling; straight pull design	Standard injection-molding process limitations	Standard injection-molding process limitations
Part material	Over 30 standard materials in stock including ABS, PC, PP, PE, Nylon 6/6, Acetal & Acrylic	Any commercially available material	Any commercially available material
Part volume	589 cm ³	No volume limit	No volume limit
Parting line geometry	No limit	No limit	No limit
Draft	Draft required in CAD model	Draft required in CAD model	Draft required in CAD model
Tolerance expectations	+/- 0.1mm or per SPE standards for material	Tighter tolerances are possible	Tighter tolerances are possible
Surface finish	Choose from 6 standard finishes	Any finish, including acid-etched finish	Any finish, including acid-etched finish
Customer owns tool?	No	Yes	Yes
Method of tool manufacture	CNC-only manufacturing, aluminum-only	No manufacturing limits, aluminum and soft steels	No manufacturing limits, tooling per customer specs
Lead time	Standard: 10 days for 254 x 254 x 76.2 mm, 20 days for 508 x 508 x 76.2 mm Expedited: 5 days, depending upon your geometry	Standard: 15-20 days Expedited: 10 days, depending upon your geometry	Standard: 4-6 weeks

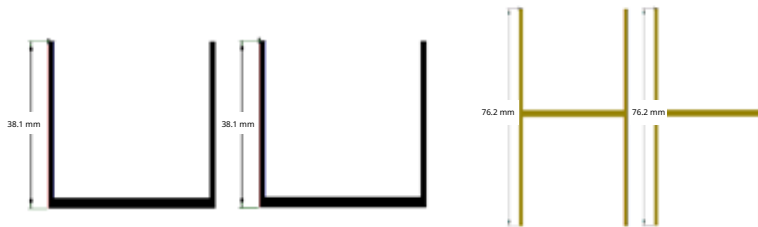
2 Size considerations

Size limits for rapid injection molding

- The XY dimensions must be less than 508 x 508 mm.
- Maximum part volume cannot exceed 589 cm³.
- Maximum part depth can be 76.2 mm, given a parting line that can pass through the middle of the part, or 38.1 mm if the parting line of the tool must be at one edge of the part.
- Parts that fall outside of this “box” can be manufactured, but must be quoted offline by your On Demand Manufacturing tooling manager.



Size Limits for rapid injection molding is 508 x 508 x 76.2 mm



The Depth Limit is 38.1 mm for parts where the parting line is on the edge of the part, and 76.2 mm for parts where the parting line is central to the part.

SIZE LIMITS FOR LOW-VOLUME INJECTION MOLDING

- The XY dimensions must be less than 914.4 x 914.4 mm.
- No volume limit.

SIZE LIMITS FOR PRODUCTION INJECTION MOLDING

- The XY dimensions must be less than 914.4 x 914.4 mm.
- No volume limit.

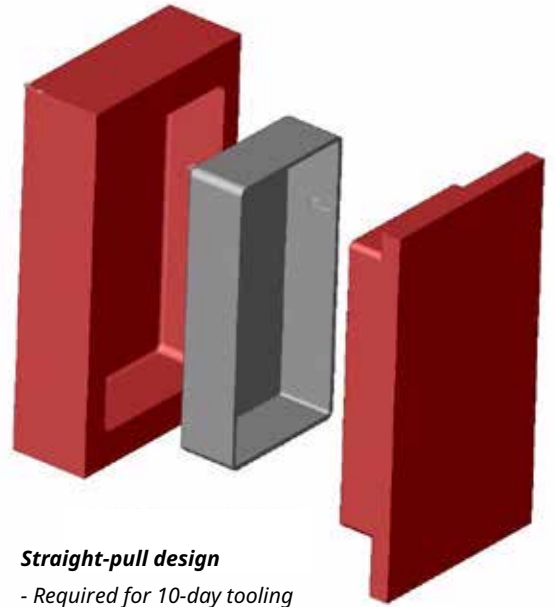
3 Considerations for undercuts

Undercut limits for rapid injection molding

Parts that qualify for rapid injection molding must be designed as straight-pull parts.

A part made with a straight-pull mold is designed such that when the two halves of the mold pull straight away from each other, there is no mold metal that wants to pass through the part plastic (an impossible, "die locked" situation).

Undercuts on the part require mold pieces to pull out sideways, perpendicular to the direction of pull. These are called side actions. Parts with undercuts are not available within the Rapid Injection Molding process.



SIZE LIMITS FOR LOW-VOLUME INJECTION MOLDING

- Undercuts are okay

SIZE LIMITS FOR PRODUCTION INJECTION MOLDING

- Undercuts are okay

4 Other geometric considerations

Parting line limitations

Rapid injection molding, low-volume injection molding and production injection molding all have no limitations on the simplicity or complexity of a part's parting line.

All 3 processes can support simple, complex, and contoured parting lines.

Deep ribs

The rapid injection molding process uses only high-speed CNC machining centers to mill out the tooling material. No special manufacturing methods, such as EDM (electric discharge machining), wire EDM or grinding, are used to manufacture the tools.

As a result of the CNC-only approach, deep ribs must be designed to accommodate this limitation.

Specifically, deep ribs require proper draft (at least 2 degrees per side) and clearance to allow the CNC machine tool to cut the root of the rib cavity.

Rule of Thumb: The maximum rib depth is 10 times the width of the rib at its smallest width. CNC cutting tool lengths (and therefore rib depths) are limited by this.

The following chart provides a rule of thumb for rib depth with 1 degree per side of draft:

Rib Depth	0.5 mm	0.75 mm	1.0 mm	1.5 mm	2.0 mm
Rib Width	3.5 mm	7.5 mm	15 mm	25 mm	36 mm

Low-volume injection molding and production injection molding have no restrictions on rib depth.




Undercuts require side action in the tool
- Not available in 10-day tooling

Rounded corners


Since rapid injection molding is a CNC-only process, sharp corners on the outside of parts (meaning sharp corners on the inside of the cavity side of the tool) are not possible.

The following illustrations show a part with sharp corners and a part with rounded corners. If your design has sharp outside corners, the CNC-only process will round off those features to a radius of 0.762 mm.



Square corners

- External square corners impossible to CNC
- Inhibits the flow of plastic, causing stress risers



Rounded corners

- External round corners are possible to CNC
- Promotes the flow of plastic for injection molding

It is strongly advised that you ensure your design has rounded corners on every rib-to-wall or wall-to-wall intersection. This design consistency will result in a part with less internal stress—reducing the chance of warp, short shots, splay and flash regardless of which injection-molding process you use.

Sharp corners

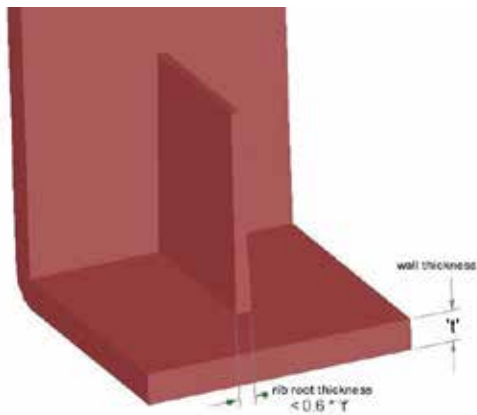
If your design requirements are so stringent that sharp corners on the outside of your part are critical to the part's function, only low-volume injection molding or production injection molding will meet that need.

Low-volume injection molding and production injection molding can accomplish the cavity-side sharp corner using EDM machining methods to literally burn the sharp corner into the tool.

Rib-to-wall thickness ratios

Thin ribs on thicker walls may provide stiffness, but they can also result in sink marks, which can be unsightly and cause tolerance problems.

Rule of thumb: Rib root thickness should equal 0.6 x wall thickness to prevent sink.



In some cases, using a glass-filled material will help avoid sink in geometries when adhering to the rule of thumb is impossible.

Warp

The flow of plastic in a tool is a complex phenomenon—sometimes friendly and predictable, other times evil and erratic.

After the molten plastic has filled the tool cavity, the plastic solidifies in the mold and freezes in a direction from the outside of the part (near the mold surface) toward the inside.

In thick sections of the part, this results in inward pulling stresses (due to the contraction of cooling), causing sink marks in the outer surfaces of the part.

Because thinner areas of the part will freeze faster than thicker sections, stresses can build up between thick and thin sections. The result is a phenomenon called “warp.”

Rule of thumb to minimize warp: Design your parts to maintain consistent wall thickness and avoid thick areas whenever possible.

Sufficient draft

The existence of draft on vertical surfaces of your part enables the easy removal of your part from the mold.

Draft is especially important in rapid injection molding since the molds are straight-pull only (i.e., no side actions) and manufactured using a CNC-only process.

The rules of thumb governing the amount of draft required (in degrees) will vary with geometry and surface texture requirements. Let's put it this way... the more draft, the better.

Rules of thumb for draft:

- Use at least 1 degree on all vertical faces
- 2 degrees works very well in most situations
- 3 degrees is a minimum for a shutoff (metal sliding on metal)
- 3 degrees is required for light and medium texture

5 Gating

Rapid injection molding tooling is created with one of the following types of gates:

- Edge gate
- Tab gate
- Center gate
- Sub gate

Gate vestige will be trimmed to ± 0.12 mm.

Low-volume injection molding and production injection molding have no restrictions on the type of gate used.

6 Resin selection

Selection of the proper material for your molded parts is a critical decision.

One must consider the mechanical properties, molding properties and cost of the resin they select for the given application. Application-specific requirements will always drive the need for particular material properties, like tensile strength and elasticity.

Successful plastic part design is based on an understanding of process-related issues during manufacturing, such as mold filling, likelihood of flash, part ejection and the potential for warp and sink.

The table below lists some commonly used resins, along with their brand names, and a high-level summary of their material properties, moldability characteristics and relative costs.

Standard rapid injection molding resins

Resin Generic Names	Resin Brand Names	Strength	Impact Resistance	Change to Stiffness
Acetal	Delrin	Medium	Medium	914.4 x 914.4 x 381 mm
Nylon 6/6	Zytel	Medium	High	No geometry limits
Nylon 6/6, glass filled	Zytel	High	Medium	Any commercially available material
Polypropylene (PP)	Marlex, Sumika	Low	High	No volume limit
Polycarbonate (PC)	Lexan	Medium	High	No limit
Acrylonitrile Butadiene Styrene (ABS)	Cycolac	Med-Low	High	Draft required in CAD model
Polycarbonate / ABS alloy	Cycology	Medium	High	Tighter tolerances are possible
Polyetherimide (PEI)	Ultem	High	Medium	Any finish, including acid-etched finish
Polyetherimide, fiber reinforced (PEI)	Ultem	Very high	Medium	Yes
Polybutylene Terephthalate (PBT)	Valox	Medium	High	No manufacturing limits, tooling per customer specs
Polystyrene	Styron	Med-Low	Low	Standard: 4-6 weeks
Thermoplastic elastomer	Thermoplastic Polyurethane, Santoprene	Low	High	

Notes: This list is not comprehensive, nor is it constant. There are many other resins not listed here. Ask your 3D Systems On Demand Manufacturing tooling manager for material selections that are currently available.

In order to avoid project delays due to material availability, we limit material selection in rapid injection molding to the material we have on hand at the time your quote is created. Your quote worksheet will present you with the currently available materials.

When your project requires a specific engineered material, it becomes either a Low-Volume Injection Molding or Production Injection Molding project. Virtually any injection moldable material can be specified, as we will source the material per your requirements.

7 Surface finish section

When designing a part for injection molding, it is important to keep in mind the relationships between surface finish, moldability, cost and lead time.

Rapid Injection Molding offers the following standard surface finishes, listed in order from lowest to highest cost:

- Non-cosmetic: finish to rapid injection molding discretion
- SPI-C3: 320 Stone finish
- SPI-B3: 320 Grit Paper
- SPI-A3: Grade #15 Diamond Buff (+2 days extra on lead time)
- Light matte texture, MoldTech 11010 Equiv. (+2 days on lead time)
- Medium matte texture, MoldTech 11060 (+2 days extra on lead time)

** SPI (The Society of the Plastics Industry) denotes an industry-standard finish.

If the part will not be visible to the end user, a non-cosmetic finish is acceptable. But many times your design will require a cosmetic surface finish.

In these cases of cosmetic finishes, there are two key limitations to be aware of: texturing and polishing.

Texturing

Since bead blasting is a line-of-sight method, it may not be possible to texture the sides of minimally drafted ribs on a part, as the mold surfaces may be inaccessible.

Also, texturing has an adverse effect on the ability of the part to release from the mold. Drag marks may result.

Therefore, we recommend that texture be specified only on areas of the part that are drafted at least 3 degrees.

Polishing

We use manual mold polishing methods to apply the SPI-A3 finish.

Since there is not significant automation in this process, you should expect a significant cost increase, as it is labor intensive and time consuming to polish deep, narrow slots in molds.

Polishing is time intensive and may also affect the lead time for your parts.

In both low-volume injection molding and production injection Molding, any surface finish can be produced, including acid-etched finish.

8 Lead time

Standard lead time for Injection Molding service is as follows:

Process	Standard Lead Time	Fastest Lead Time
Rapid Injection Molding	10 days for 254 x 254 x 76.2 mm parts 20 days for 508 x 508 x 76.2 mm parts	5 days, depending upon geometry
Low-Volume Injection Molding	15-20 days	10 days, depending upon geometry
Production Injection Molding	4-6 weeks	4 weeks

Lead times can change based on current factory load. Call 3D Systems On Demand Manufacturing tooling manager at +61 3 9819 4422 to discuss your specific lead time requirements.