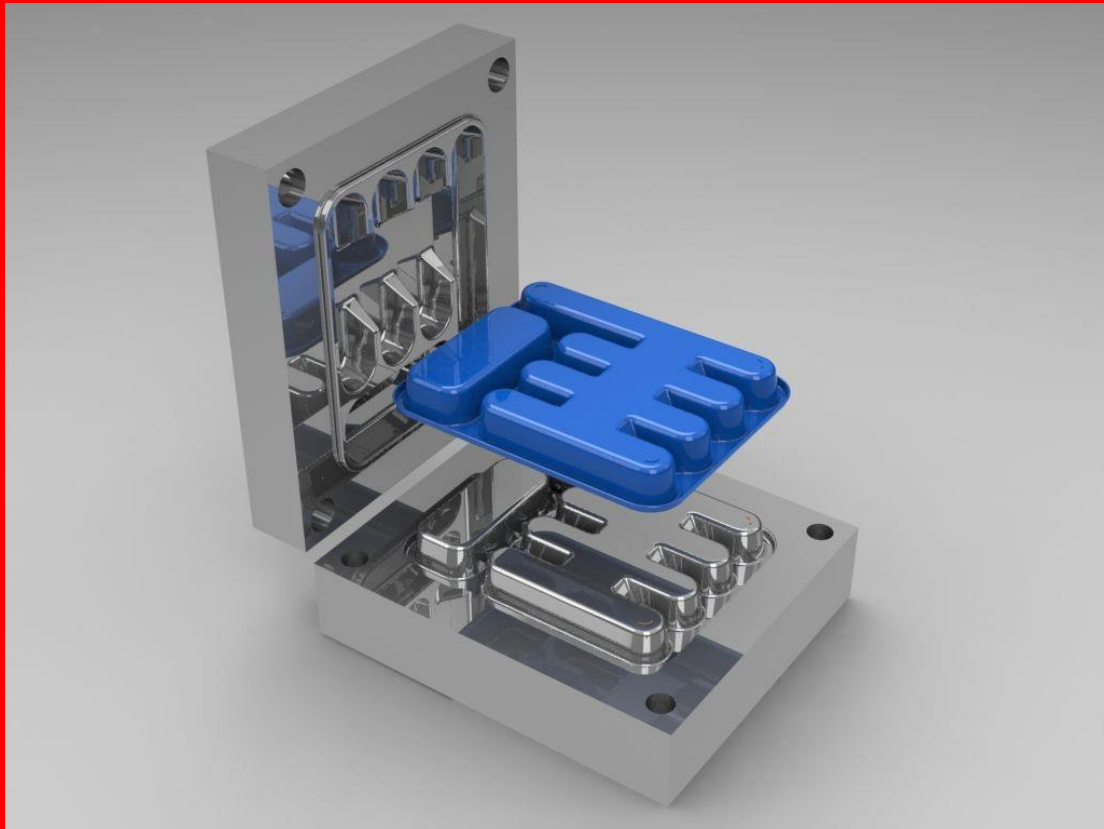


Handbook

Injection Molding Guide



For beginners and professionals

IN3DTEC V6.2.1

Contents

1 >>The basics

1.1 What is injection molding-----	Page 4
1.2 Overview of injection molding process -----	Page 4-5
1.3 Pros and cons of injection molding -----	Page 5-6
1.4 The applications of injection molding-----	Page 6-9

2 >> Design tips for injection molding

2.1 Draft -----	Page 9-11
2.2 Wall thickness-----	Page11-15
2.3 Radii -----	Page15-16
2.4 Thread features -----	Page16-17
2.5 Living hinges -----	Page17
2.6 Lettering and symbols -----	Page17
2.7 Undercuts -----	Page17-19

3 >> Materials and surface finish for injection molding

3.1 10 commonly used injection molding materials -----	Page19-24
3.2 Surface finish -----	Page24-25

4 >> Cost reduction tips

4.1 Optimized plastic parts design -----	Page25-26
4.2 Don'ts use unnecessary cosmetics and features-----	Page26
4.3 Material selection-----	Page26

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

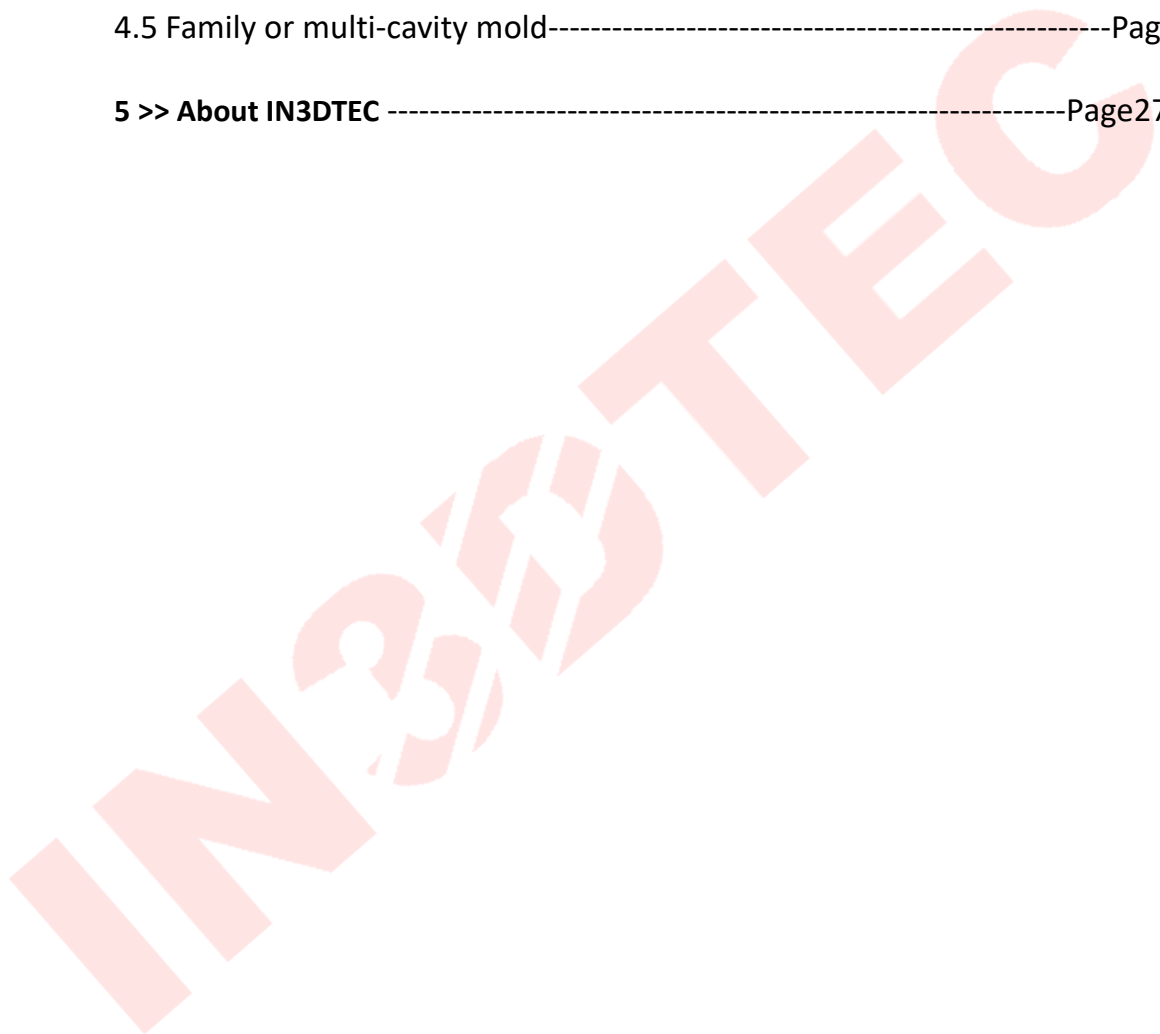
Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

4.4 Review your part tolerances -----Page27

4.5 Family or multi-cavity mold-----Page27

5 >> About IN3DTEC -----Page27-28



1 >> The basics

1.1 What is injection molding

Injection molding is a manufacturing technology for the mass-production of identical plastic parts with good tolerances. In Injection Molding, polymer granules are first melted and then injected under pressure into a mold, where the liquid plastic cools and solidifies. The materials used in Injection Molding are thermoplastic polymers that can be colored or filled with other additives.

Almost every plastic part around you was manufactured using injection molding: from car parts, to electronic enclosures, and to kitchen appliances.

Injection molding is so popular, because of the dramatically low cost per unit when manufacturing high volumes. Injection molding offers high repeatability and good design flexibility. The main restrictions on Injection Molding usually come down to economics, as high initial investment for the mold is required. Also, the turn-around time from design to production is slow (at least 4 weeks).

1.2 Overview of injection molding process

The first stage of injection molding is to create the mold itself. Most molds are made from metal, usually aluminum or steel, and precision machined to match the features of the product they are to produce.

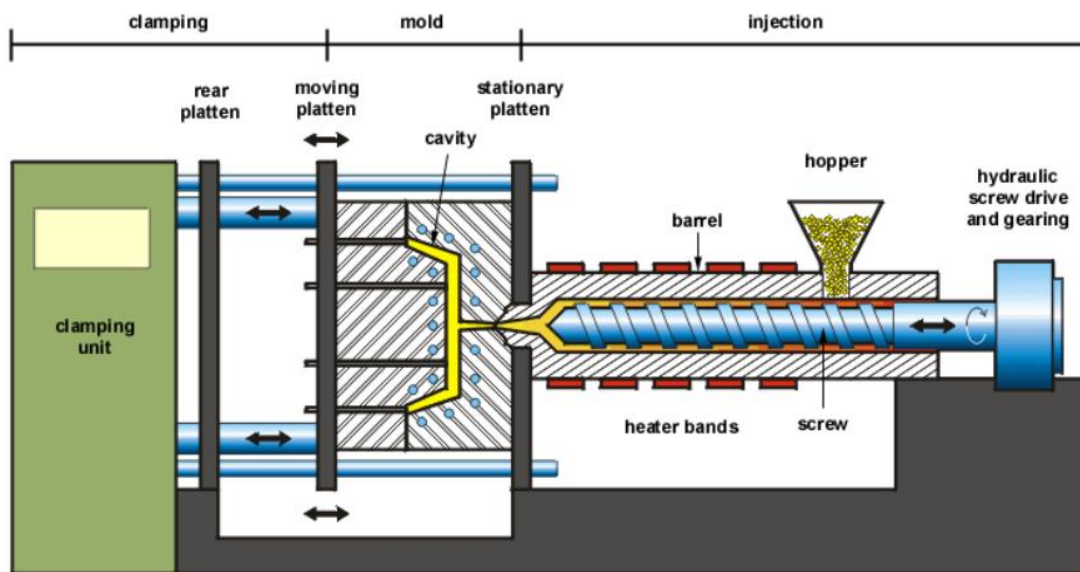
Once the mold has been created by the mold-maker, the material for the part is fed into a heated barrel and mixed using a helical shaped screw. Heating bands melt the material in the barrel and the molten metal or molten plastic material is then fed into the mold cavity where it cools and hardens, matching the shape of the mold. The cooling time can be reduced through the use of cooling lines that circulate water or oil from an external temperature controller. Mold tools are mounted on plate molds (or 'platens'), which open once the material has solidified so that ejector pins can eject the part from the mold.

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com



1.3 Pros and cons of the injection molding

In this guide, we will focus on CNC machines that remove material using cutting tools. These are the most common and have the widest range of applications. Other CNC machines include laser cutters, plasma cutters and EDM machines.

Pros

low cost per unit when manufacturing high volumes.

The main advantage of injection molding is being able to scale up production to produce a large number of parts. Once the initial costs of the design and the molds have been covered, the price of manufacturing is very low. The cost of production drops as more parts are produced.

High productivity

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

Great reliability and consistency

injection molding allows for the production of many identical parts, which allows for part reliability and consistency in high volume production.

Wide range of materials

Almost every thermoplastic material (and some thermosets and silicones) can be injection

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |

Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

molded. This gives a very wide range of available materials with diverse physical properties to design with.

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

Excellent visual appearance

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

Cons

High up-front cost for injection tool

Up-front costs can be high for injection molding, particularly with regard to tooling. Before you can produce any parts, a prototype part needs to be created. Once this has been completed, a prototype mold tool needs to be created and tested. This all takes time and money to complete and can be a costly process.

Design changes are costly

After a mold is manufactured, it's very expensive to modify. Design changes usually require the creation of a new mold from scratch. For this reason, correctly designing a part for injection molding is very important.

Longer lead times than other technologies

The typical turnaround for injection molding varies between 6-10 weeks. 4-6 weeks to manufacture the mold, plus 2-4 more weeks for production and shipping. If design changes are required (something quite common) the turnaround time increases accordingly.

In comparison, parts made in a desktop 3D printer can be ready for delivery overnight, while industrial 3D printing systems have a typical lead time of 3-5 days. CNC machined parts are typically delivered within 10 days or as fast as 5 days.

1.4 The applications of injection molding

Injection molding is the preferred production method for most mass-produced plastic products, due to its high output rate and consistency of quality. These products include but are not limited to:

Automotive parts – dashboards, bumpers, grilles.

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com



Electronic components – electrical connectors, enclosures, protective sleeving.



Medical devices – syringes, valves, dishes.



Consumer plastics – mobile phone cases, bottle caps, toys.



Furniture parts – seat cushions, chairs, seat covers.

IN3DTEC TECHNOLOGY CO., LTD.
3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design
Digital manufacturing services for prototyping & on-demand production
info@in3dtec.com; www.in3dtec.com



2 >> Design tips for injection molding

Injection molding is a formative manufacturing technology, i.e. material is formed from an amorphous shape into a fixed shape defined by a mold tool. Almost every plastic part created today is by injection molding as it allows identical parts to be created in huge numbers, in a short space of time, and at very low cost per part.

Best practice design guides aim to help create complex shapes while:

- Allowing plastic to flow easily and uniformly around the part.
- Allowing the plastic to cool quickly and evenly, resulting in a stable and accurate part.

These general tips will improve part quality, mold ability and cycle time based on known implementation and characteristics of the injection molding process.

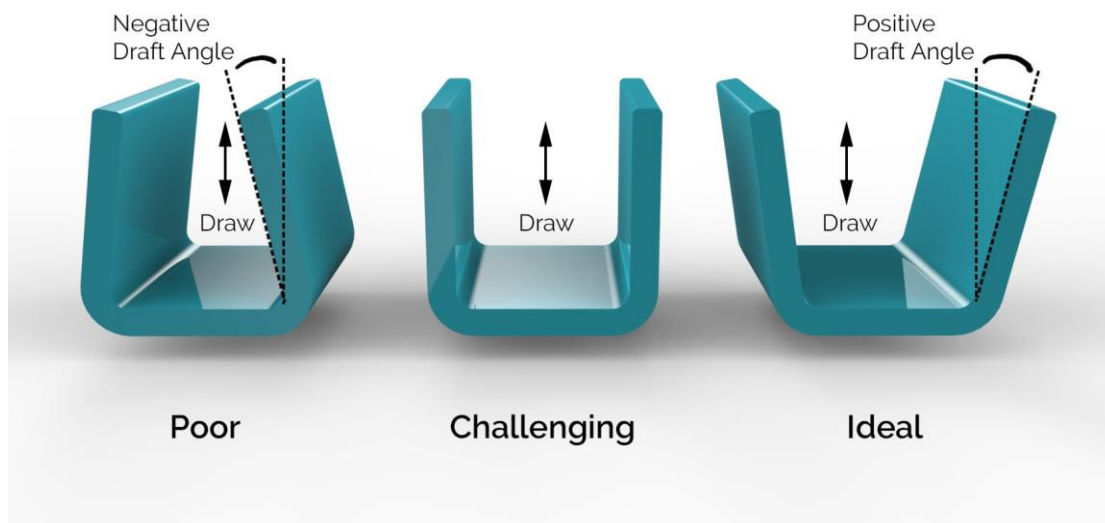
2.1 Draft

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com



Draft, or the application of a slight taper to every surface in the direction of pull on an injection-molded part, is a small and even tedious design element — but one that’s vital to the success of a project. To visualize draft, envision an ice cube tray: the slight taper allows ice cubes to slide out easily without falling victim to excessive suction or friction. Parts that are lacking the appropriate amount of draft — or a suitable draft substitute — will not properly eject from the mold.

What’s more, draft protects the part from damaging friction, reduces wear and tear during the ejection process, helps ensure a uniform finish, and reduces costs by avoiding the need for complex injection setups. Fortunately, no toolmaker would make a part without draft. For that reason, designing for optimized draft angles doesn’t just mean adding draft; in most cases, draft is a given. Rather, optimizing draft means carefully incorporating draft so that it adds to, rather than interferes with, the design and look of the final part.

The minimum draft angle for any given part is largely driven by the depth of draw, the wall thickness, the material’s shrink rate, and the surface finish or texture that is to be applied. As a general rule, a draft angle of 1.5 to 2 degrees is required for most parts, but draft should average about an additional degree for each extra inch of part depth. Note that if a part is very small, there’s some more flexibility to decrease draft below 1.5 degrees. However, for most parts, 1.5 degrees is the minimum draft requirement.

That said, texture also plays an important role in determining draft. Many injection-molded parts have a leather grain or other texture applied to their surface for aesthetic purposes; however, depending on how deep the texture is, the draft angle may need to be increased to ensure the texture won’t be scraped off or damaged during the ejection process.

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

Automobile interiors are a strong example of strategically-applied draft. Most modern automobile interiors are injection-molded but feature a leather grain texture; a careful eye can discern that the texture depth varies throughout the part in order to accommodate changing draft, but it's barely noticeable. On the other hand, many cheaply-made consumer goods have visibly different textures throughout the part or even texture that has been noticeably scraped off.

2.2 Wall thickness

If you take apart any of the plastic appliances around your home (as most engineers probably did as children) you'll notice that the walls for most parts are about 1 mm to 4 mm thick (the optimal thickness for molding), and uniform for the entire piece. Why? Two reasons.

First of all, thinner walls cool faster, shortening the cycle time of the mold, the amount of time it takes to make each part. If a plastic part can cool faster after the mold is filled, then it can safely be ejected sooner without warping, and because time on the injection machine costs money, the part is less expensive to produce.

The second reason is uniformity: In the cooling cycle, the outer surface of a plastic part cools first. Cooling causes contraction; if the part is of uniform thickness, then the entire part will shrink away from the mold uniformly as it cools, and the part comes out smoothly.

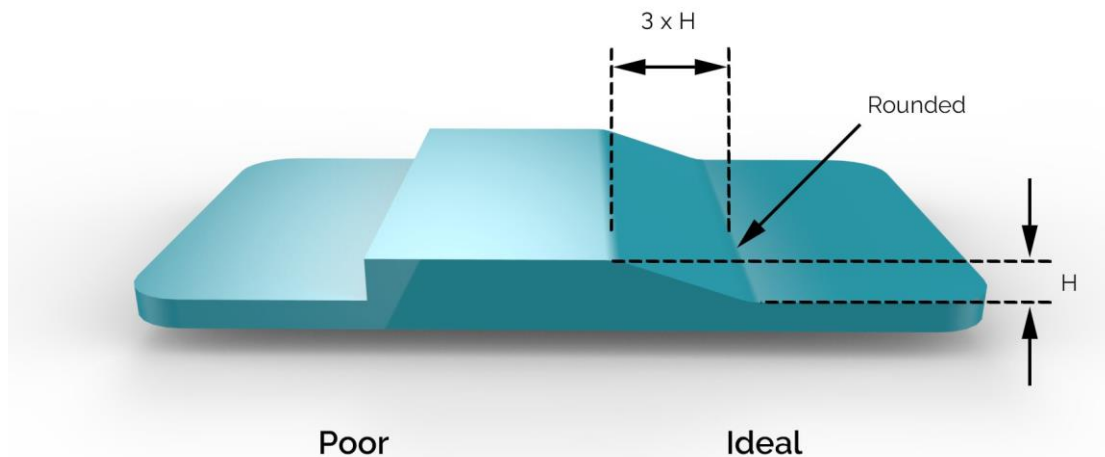
However, if the part has thick and thin sections next to each other, then the molten center of the thicker area will continue to cool and contract after the thin areas and surfaces have already solidified. As this thick area continues cooling, it keeps contracting, and it can only pull material from the surface. The result is a little dimple on the surface of the part called a sink mark.

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

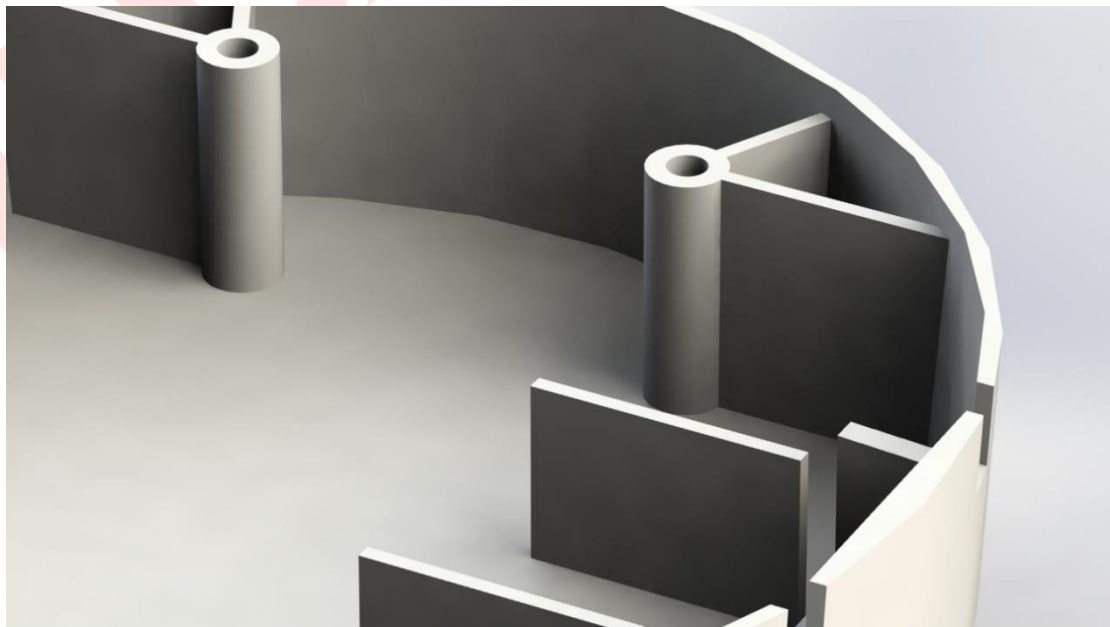
Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com



Happily, thick walls have some simple solutions. The first thing to do is to notice the areas which are a problem. In the part below, you can see two common issues: thickness around screw holes, and thickness where strength is needed in the part.

For screw holes in an injection molded part, the solution is to use “screw bosses”: a small cylinder of material directly around the screw hole, tied to the rest of the housing using a rib or a flange of material. This allows for more uniform wall thickness and fewer sink marks.



When an area of the part needs to be especially strong, but the wall is too thick, the solution is similarly simple: ribbing. Instead of making the entire part thick and difficult to cool, thin the exterior face into a shell, then add vertical ribs of material to the interior for strength and rigidity.

IN3DTEC TECHNOLOGY CO., LTD.

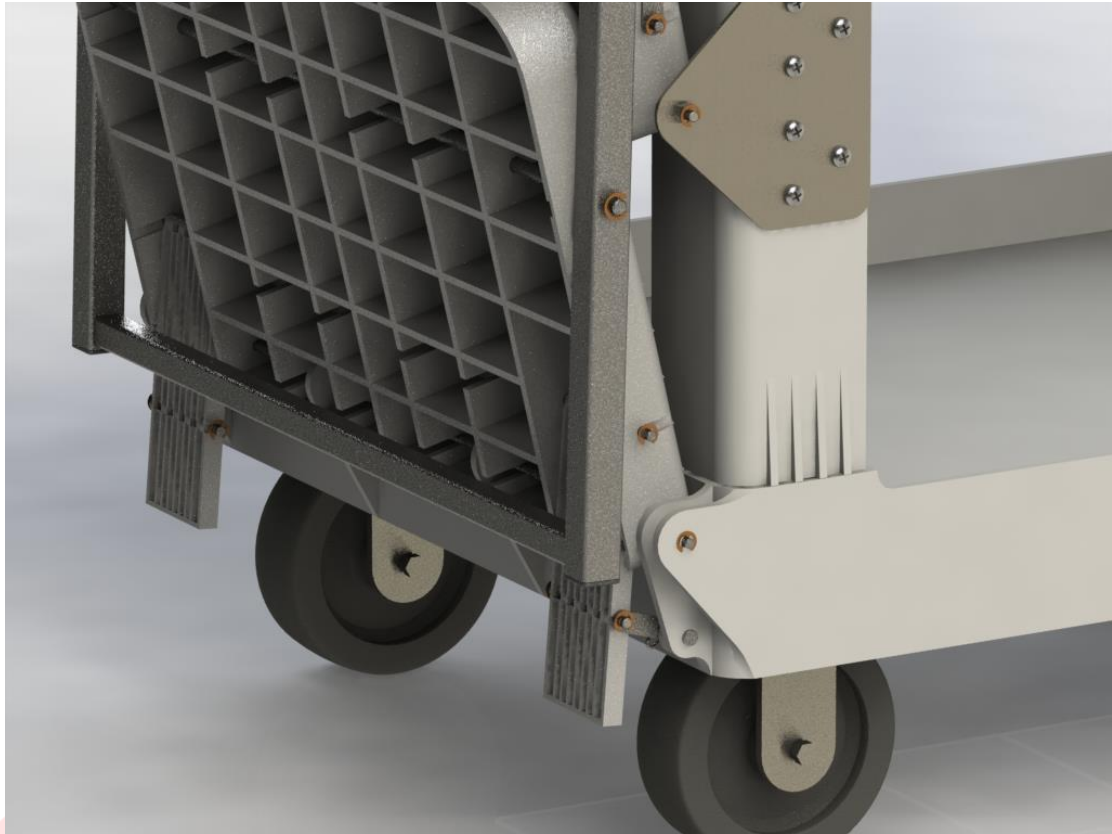
3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |

Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

In addition to easier molding, this reduces the amount of required material, reducing costs.



Guidelines for rib design are as follows:

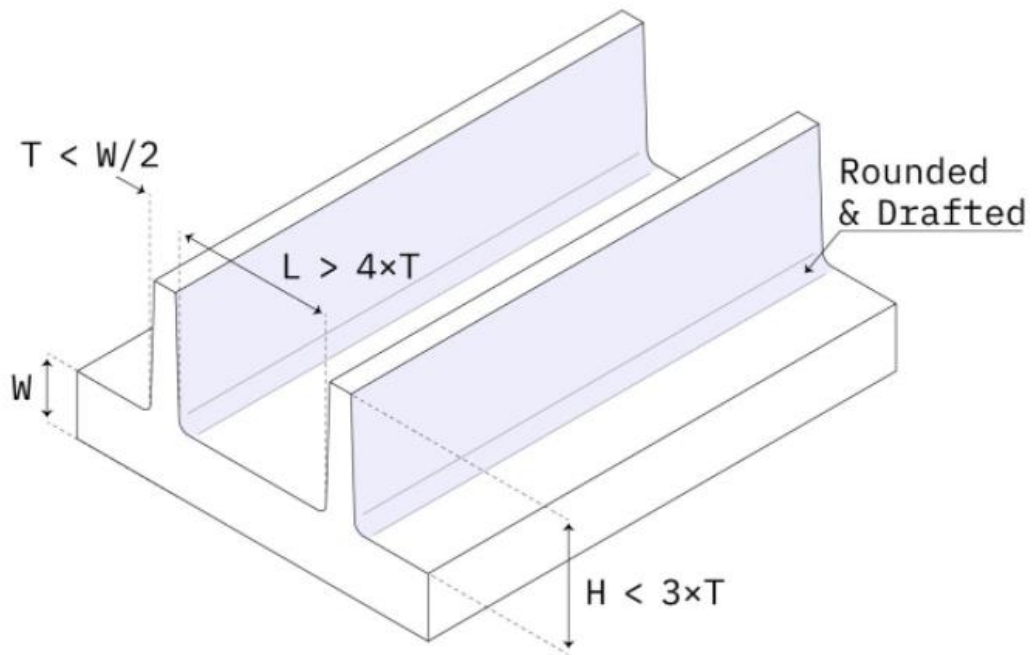
- Ribs should have a maximum thickness of 0.5 times the wall thickness.
- Ribs should have a maximum height of 3 times the wall thickness.
- Ribs should have rounded edges to help avoid sink marks.
- Add a draft angle of at least 0.25° - 0.5°
- Add a min. distance between ribs and walls of $4 \times$ rib thickness

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com



The following are recommended wall thickness for different materials:

Material	Recommended Wall Thickness (mm)
General Rule of Thumb	1.2 - 3.0
ABS	1.1 - 3.5
Acetal (POM)	0.6 - 3.8
Liquid Crystal Polymer	0.75 - 3.0
Long-Fibre Reinforced Plastics	1.9 - 25.4
Nylon (PA 6)	0.75 - 3.0
Polycarbonate (PC)	1.0 - 4.0
Polyester	0.6 - 3.2
Polyethylene (PE)	0.75 - 5.0
Polyethylene Sulfide	0.5 - 4.5
Polypropylene (PP)	0.6 - 3.8
Polystyrene (PS)	0.9 - 4.0
Polyurethane (PUR)	2.0 - 20
PC/ABS	1.2 - 3.5
Silicon	1.0 - 10

2.3 Radii

Correct placement of corner radii in injection molding design creates strong, high-quality and cost-effective plastic parts. Sharp external corners are ok and sometimes necessary to fulfill product requirements, such as triangular shaped items. However sharp corners can present challenges when designing for injection molding, as they can cause stress resulting in a poor product, radii are key to reducing stress.

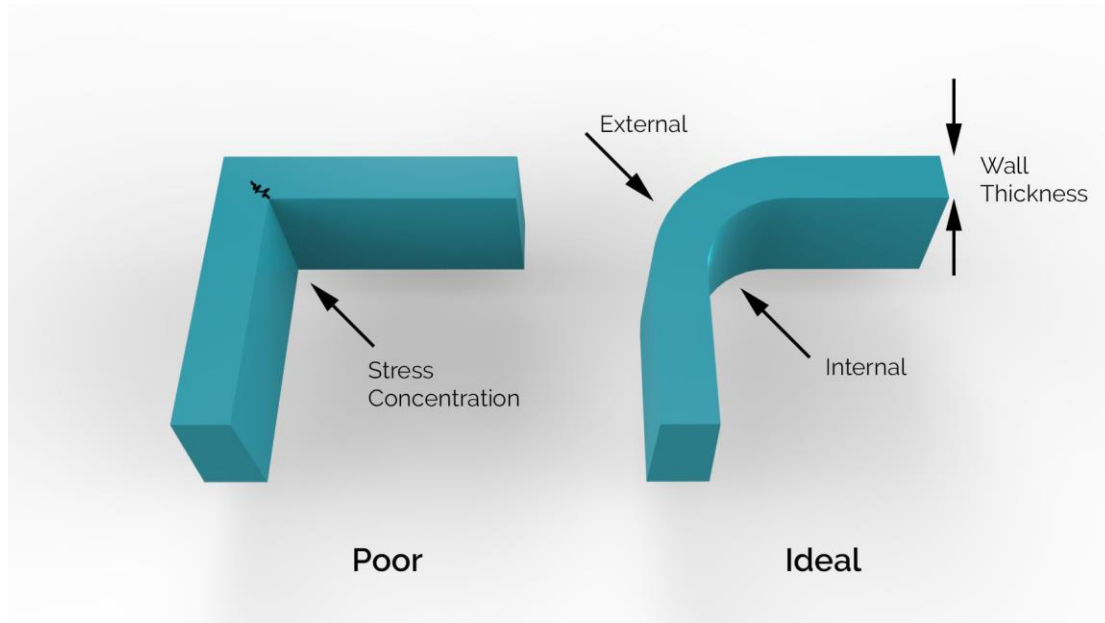
There are two types of radius, internal and external. Internal edges should be rounded to a minimum of 0.5 times the wall thickness. External edges should be rounded to a minimum of 1.5 times the wall thickness.

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com



2.4 Thread features

There are two main ways to add a thread into an injection molded part. Each method has its merits and is suited to different applications.

Molded Thread

A molded thread is a thread molded directly into the tool (an example of this is a bottle lid). Because the thread creates undercuts, then in order to remove the part from the tool, the thread will need to be unscrewed. For this reason, the tool will need to be more complex and costly.

Generally, this will be best suited to larger threads on a simpler overall part.

Bosses

Bosses are very common method of creating an attachment point in injection molding. They are simply cylindrical extrusions that can accommodate a self-tapping screw, metal threaded insert or feature from another part.

Metal threaded inserts can be added into the boss by ultrasonic, thermal or in-mold insertion. These allow machine threads and are well suited to higher load applications or which require many cycles of assembly and disassembly.

Best design practice for bosses are as follows:

-Outside diameter 2 times the internal diameter.

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

- Add chamfer to guide screw or insert into hole.
- The hole should extend to the wall level.

2.5 Living hinges

Living hinges are thin sections of plastic that connect two segments of a part to keep them together and allow the part to “hinge” open and closed. Typically these hinges are incorporated in containers that are used in high volume applications such as toolboxes and CD cases.

Materials used in molding living hinges must be very flexible, such as polypropylene or polyethylene. A well-designed living hinge typically flexes more than a million cycles without failure.

2.6 Lettering and symbols

Text is a very common feature that can be useful for logos, labels, warnings, diagrams and instructions, saving the expense of stick-on or painted labels.

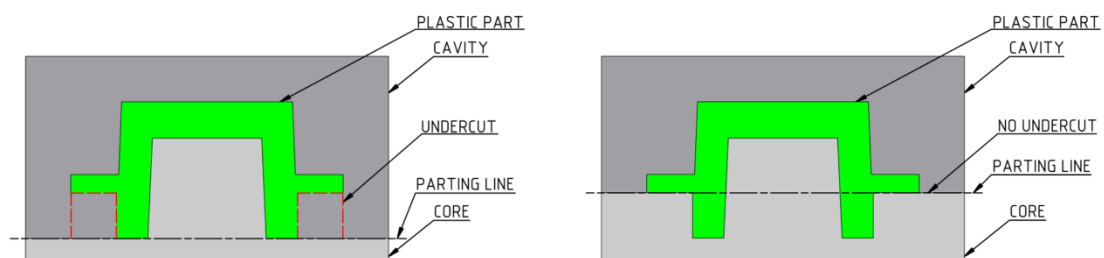
When adding text, choose embossed text over engraved text, as it's easier to CNC machine on the mold and thus more economical.

Also raising the text 0.5 mm above the part surface will ensure that the letters are easy to read. We recommend selecting a bold, rounded font style with uniform line thickness, with a size of 20 points or larger. Some font examples include: Century Gothic Bold, Arial and Verdana.

2.7 Undercuts

Achieving success with undercuts requires minor mold modifications and a lot of expertise. Some of the designs that can help avoid defects and wearing of the mold include:

Parting Lines: By moving the parting line and adjusting draft angles to intersect an undercut, you can prevent part defects. The parting line placement is limited according to the geometry, material flow, and other features of the part.



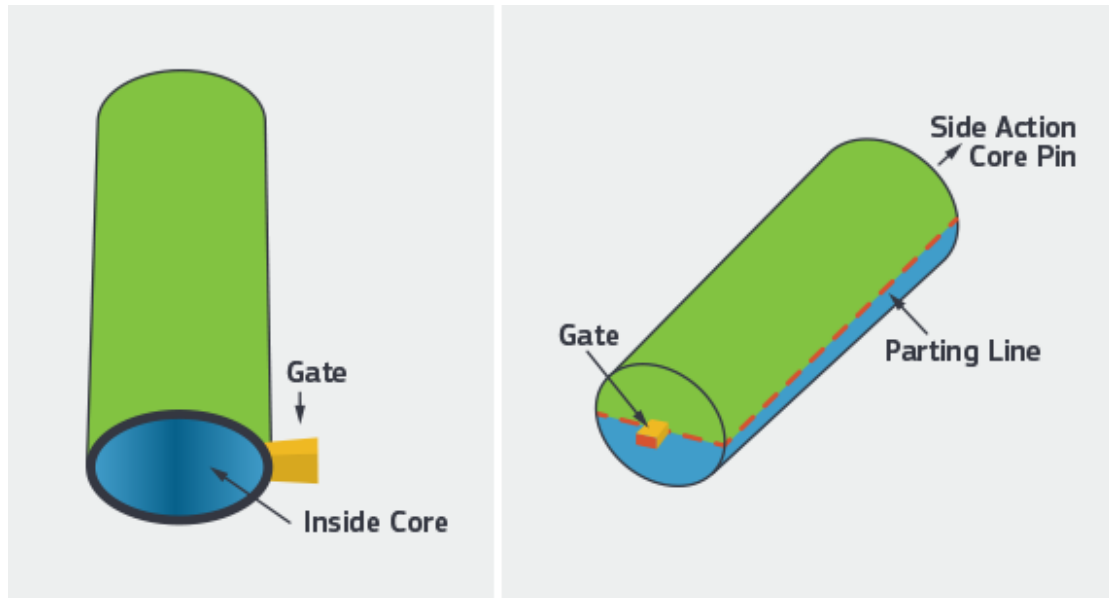
IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

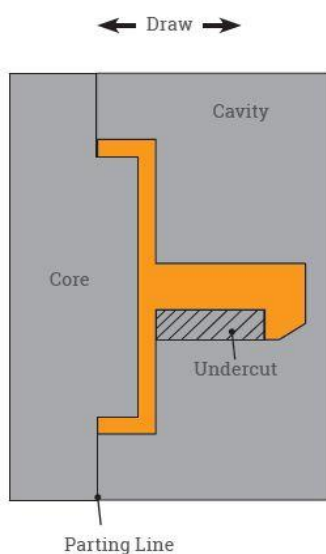
Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

Side-Actions: A perpendicular side-action is ideal for cylindrical parts, as the mold is split horizontally along the part. After the resin is shot into the mold and begins to cool, the side-action slides on an angled pin until its clear from the undercut, allowing it to be freely ejected.



Sliding Shutoffs: This technique uses create clip- and hook-style components to lock together two halves of a mold. During mold operation, these mechanisms seal together, “shutting off” certain areas of the part to create complex features, such as holes.



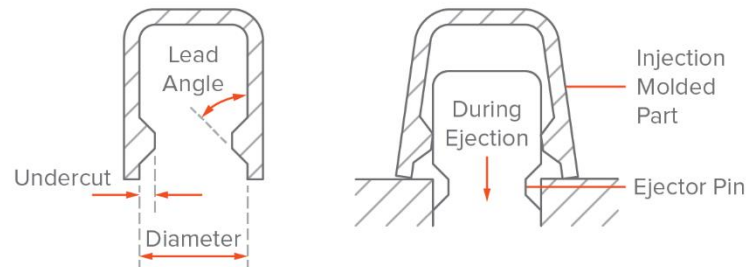
IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

Bump Offs: If you have a mild undercut, you can make a separate insert that bolts into the mold. Upon ejection, the plastic briefly stretches over the insert but subsequently resumes its desired form.



Hand-loaded Inserts: A machined insert is hand-loaded into the mold to prevent molten plastic from flowing into these areas. Upon completion of the cycle, the inserts are ejected with the part, where an operator is required to pick them off the part for further use.

3 >> Materials and surface finish for injection molding

Material selection is an incredibly important part of any manufacturing process, but engineers and product teams must be especially diligent when it comes to selecting materials for CNC machining. Since this process is compatible with a wide variety of materials — from metal to plastic — it's easy to mistakenly choose a material that's sufficient for the project but not the best-suited. In this article, we'll touch on key considerations for CNC machining material selection and give an overview of some common options.

3.1 10 commonly used injection molding materials

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

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

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

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

Polypropylene (PP)

This thermoplastic material is widely used in the food storage and packaging industry, as it does not let chemicals mix with food products. Polypropylene can be washed in hot water without degrading, and it has high chemical and moisture resistance. It has incredible impact strength, elasticity and toughness.

Acrylonitrile Butadiene Styrene (ABS)



This is an opaque thermoplastic polymer and an engineering grade plastic. There are many advantages to using Acrylonitrile Butadiene Styrene. It is tough, has good dimensional stability, high impact resistance and resistance to scratching, breaking and tearing. It has low melting temperature and is relatively inexpensive. It is commonly used to produce electronic parts such as phone adaptors, keyboard keys and wall socket plastic guards.

Polyethylene (PE)

Polyethylene is a lightweight thermoplastic material that has high chemical resistance, elasticity and electrical insulating properties. It is not especially strong or hard, but still, this material is used in wide variety of consumer plastic parts, such as milk bottles, medicine and detergent bottles, plastic bags and trash cans. Polyethylene is also the most common injection molding plastic type to be used in the production of children's toys.

Polystyrene (PS)

When it comes to injection molding, there are two types of polystyrene that are commonly used. High Impact Polystyrene (HIPS) and General Purpose Polystyrene (GPPS). General Purpose Polystyrene is transparent, while High Impact Polystyrene is opaque. Hard cases for toolboxes and bodies of power tools are also made using High Impact Polystyrene. While they are

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |

Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

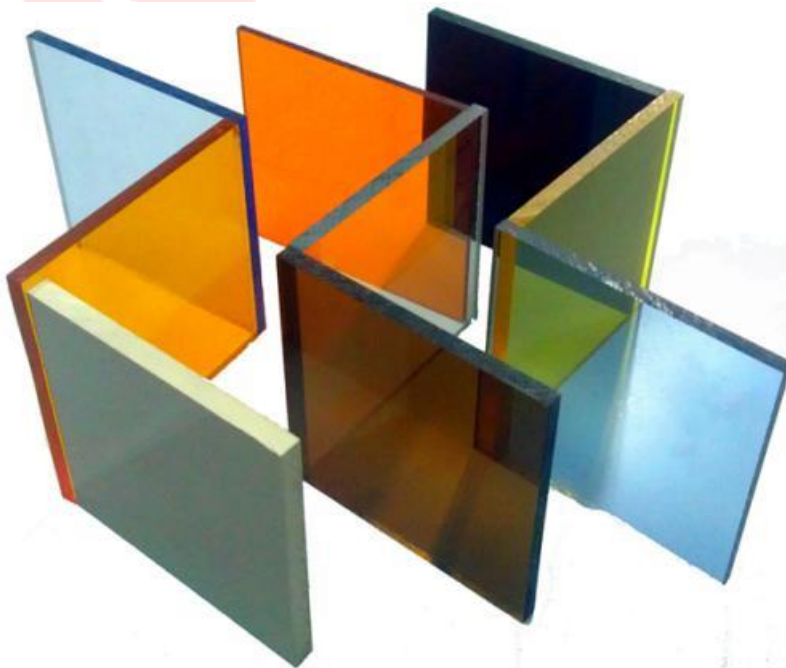
inexpensive, they are not environmentally friendly.

Nylon (PA)



Nylon is often used to produce strong mechanical parts. Bushings, gears and bearings are some common automotive parts produced using nylon. It is tough, and has a high melting temperature, good wear and chemical resistance. Nylon is also known by its chemical designation PA (Polyamide).

Acrylic



IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

We use acrylic to produce transparent parts such as windows, transparent walls and various lighting equipment. It is often used as an alternative to glass due to its high tensile strength and weather and scratch resistant nature. On top of its optical and transparent properties, it also has many unique qualities. You would usually spot this material in refrigerators and containers to store food, as acrylic is odorless and tasteless and doesn't contain Bisphenol A (BPA), a potentially harmful organic synthetic compound that's not suitable for food preparation and storage.

Polycarbonate (PC)

Polycarbonate has excellent optical properties and is extremely durable. When molding with this amorphous thermoplastic material, precise dimensional control can be maintained as it has predictable and uniform mold shrinkage. We use polycarbonate when we need something substantially stronger than acrylic. This material is not suitable for food storage and preparation as it contains Bisphenol A (BPA).

Polyoxymethylene (POM)



Polyoxymethylene is a type of acetal resin used for producing mechanical and automotive parts that would usually be made with metal. This engineering thermoplastic material is very strong, tough and rigid. It is often used to produce gears, fasteners, knife handles and ball bearings. Although polyoxymethylene has high resistance towards solvents such as alcohols, gasoline, detergents and motor oils, it should not be exposed to hydrochloric acid and nitric acid.

Thermoplastic Polyurethane (TPU)

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com



Thermoplastic polyurethane, because of its soft and elastic nature and great tensile and tear strength, is often used to produce parts that demand a rubber-like elasticity. TPU performs well at high temperatures and is commonly used in power tools, cable insulations and sporting goods.

Thermoplastic Rubber (TPR)

















This material is usually a mix of plastic and rubber, and is easy to use in the injection molding process. TPR exhibits properties of both rubber and plastic materials. It has outstanding chemical and weather resistance and high impact strength. You can find this recyclable material in medical catheters, suspension bushings and headphone cables. Thermoplastic rubber is also known as thermoplastic elastomer (TPE).

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

RECYCLING CODES FOR PLASTIC						
						
PETE or PET	HDPE or PE-HD	PVC or V	LDPE or PE-LD	PP	PS	0 or N/A
Polyethylene Terephthalate	High-Density Polyethylene	Polyvinyl Chloride	Low-Density Polyethylene	Polypropylene	Polystyrene or Styrofoam	Other
						
soda bottles, water bottles, polyester film, containers for food, jars, fibers for clothing	detergent containers, plastic bottles, piping for water and sewer, snowboards, boats	window frames, plumbing products, electrical cable insulation, clothing, medical tubing	shopping bags, plastic bags, clear food containers, disposable packaging	laboratory equipment, automotive parts, medical devices, food containers	CD and DVD cases, packing peanuts, single-use disposable cutlery, trays	baby feeding bottles, car parts, water cooler bottles, sippy cups

3.2 Surface finish

Surface finishes can significantly affect the look of the finished part. They can also affect the mechanical properties of the part as friction will increase as average surface finish (Ra) increases. This friction will also lead to increased wear on the tool and so more generous draft angles maybe required.

Finish	SPI Standard	Finishing Method	Typical Surface Roughness Ra (µm)
Super High Glossy	A-1	Grade #3, 6000 Grit Diamond Buff	0.012 – 0.025
High Gloss	A-2	Grade #6, 3000 Grit Diamond Buff	0.025 – 0.05
Normal Glossy	A-3	Grade #15, 1200 Grit Diamond Buff	0.05 – 0.10
Fine Semi-Glossy	B-1	600 Grit Paper	0.05 – 0.10
Medium Semi-Glossy	B-2	400 Grit Paper	0.10 – 0.15
Normal Semi-Glossy	B-3	320 Grit Paper	0.28 – 0.32
Fine Matte	C-1	600 Grit Stone	0.35 – 0.40
Medium Matte	C-2	400 Grit Stone	0.45 – 0.55
Normal Matt	C-3	320 Grit Stone	0.63 – 0.70
Satin Textured	D-1	Dry Blast Glass Bead #11	0.80 – 1.00
Dull Textured	D-2	Dry Blast #240 Oxide	1.00 – 2.80
Rough Textured	D-3	Dry Blast #24 Oxide	3.20 – 18.0
As Machined	-	Finished to the Machinist's Discretion	3.20 (with visible machining marks)

4 >> Cost reduction tips

While it is now regarded as a fairly cost-effective process, injection molding is fundamentally a higher cost manufacturing route. However, the advantages that it offers in terms of precision, stability, fast throughput and repeatability make it desirable. If you are considering using it as a production method, but think that it doesn't quite fit your manufacturing model, perhaps you should look at how it can be modified to reduce costs without skimping on all of the really desirable attributes.

The injection molding process is actually a lot more flexible than you may think, and there are many elements of it which can be tweaked to reduce both the costs of the molds and the products that they are used to make. With a little thought and some collaboration with your mold designer and manufacturer, you could soon see injection molding as a viable process for even fairly low-number runs.

4.1 Optimized Plastic Parts Design

Complex part designs with intricate geometries require complex molds to execute the project.

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

Mold designs usually have 2 sides: sides A and B. Side A, also known as the cosmetic side, is usually the side seen by the user. Side A is expected to be smooth and aesthetically pleasing. Side B contains the hidden structures that support the usage of the part. Side B structure may include ribs, bosses, etc., and its finish is usually much rougher than that of Side A. Molds with complex Side A and Side B designs are generally more expensive to fabricate compared to simpler molds. Complex designs that feature undercuts might also require sliding side-actions and cores, increasing the cost of the mold.

Eliminate undercuts

Undercuts require more complex tooling such as sliding cores. Eliminating undercuts can lower tooling costs and avoids additional time being spent working on the mold. For example, the clip feature on the left below is complex to injection mold due to the undercut present. By adding a slot underneath the clip feature, the undercut is removed and the part can be tooled far more easily.

Ribs help minimizing the part volume

When an area of the part needs to be especially strong, but the wall is too thick, the solution is similarly simple: ribbing. Instead of making the entire part thick and difficult to cool, thin the exterior face into a shell, then add vertical ribs of material to the interior for strength and rigidity. In addition to easier molding, this reduces the amount of required material, reducing costs.

4.2 Don't use unnecessary cosmetics and features

While you shouldn't skimp on your features and cosmetics by any means, this doesn't mean you should add every bell and whistle you can think of either. It's nice to have a very aesthetically pleasing part, but the extra effort required in the mold you'll be using can sometimes cut deeply into your profit margin.

For instance, if you want extra features on your part, you may need extra mold work to be done, such as EDM, bead blasting, or high mold polishing to achieve the specific cosmetic appearance you are seeking. This extra work can drastically impact cost reduction efforts.

At a smaller level, including company logos, textured surfaces, and molded part numbers are all features that, if deemed unnecessary, are penny pinchers that can help when pricing out injection molding costs

4.3 Material selection

Unless you're designing a part for use in harsh environments such as extreme hot or cold, or for special-grade uses such as medical or food, material choice is often interchangeable. Rarely will you need to select a "Cadillac"-grade resin for a general-use part; and picking a lower cost material that can still suit your needs is an easy and effective way to lower your overall costs. A simple analysis of use cases for your product, along with quality requirements and your target market, can help you choose the right material for your price point.

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com

4.4 Review your part tolerance

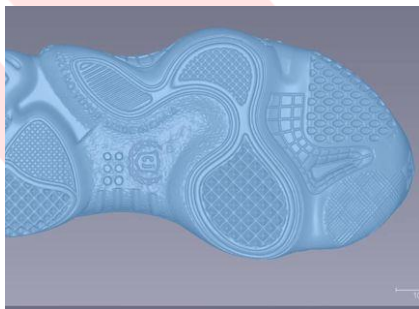
Specifying overly tight tolerances will require additional mold manufacturing steps, increasing the mold manufacturing cost, production and molding cost, as well as residual maintenance cost. Ensure the tolerances you specify are truly necessary to meet your needs and identify where there's room for flexibility. For some manufacturers, however, tight tolerances are justifiable to ensure part consistency over the long term. For example, Lego® building blocks have tolerances as small as 10 micrometers because they must engage with every other Lego block made over the last half-century, fitting firmly, yet allowing a child to disassemble them easily.

4.5 Family or Multi-cavity mold

Another way to reduce costs related to tooling is designing a family or multi-cavity mold. Depending on part similarity, material selection, and processing requirements you may be able to include more than one part in your mold. This will increase the cost of the tool but it will be marginal when compared to cutting separate tooling for each part.

Multi-cavity molding lends itself to high production requirements. The more cavities you have in a tool the more parts you're able to create each cycle. While this will drive up initial costs related to tooling it will lead to lower part prices and lower cost overall in the long run.

5 >> About IN3DTEC Our services



[3D Scanning & Reverse engineering>>](#)



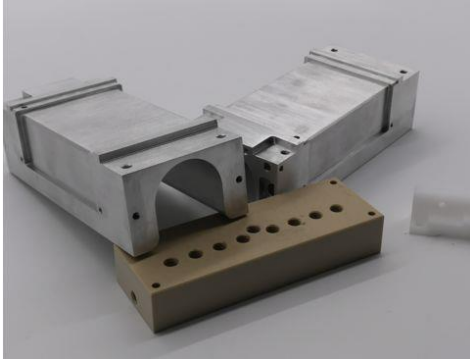
[3D Printing Service>>](#)

IN3DTEC TECHNOLOGY CO., LTD.

3D Scanning & Reverse Engineering | 3D Printing | CNC Machining | Vacuum Casting | Injection Molding |
Product Design

Digital manufacturing services for prototyping & on-demand production

info@in3dtec.com; www.in3dtec.com



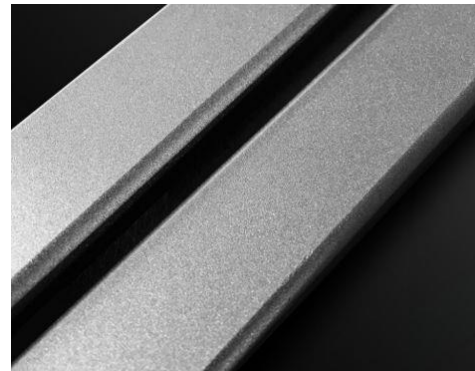
[CNC Machining Service>>](#)



[Vacuum Casting Service>>](#)



[Injection Molding Service>>](#)



[Surface finishes Service>>](#)

Thank you very much for taking the time to read this manual, and we welcome your comments or suggestions.

Attention: The information contained in this document is for the exclusive use of intended addressee(s)/receivers only and is confidential information. If you have received this copy in error, please reply to the sender highlighting the error and destroy the original message and all copies immediately. Thanks.

IN3DTEC

Bring your concept into reality

IN3DTEC was established in 2014 by a professional team in Shanghai with branches in Hong Kong, Suzhou, Shenzhen. We provide a full spectrum of technologies including 3D scanning, 3D printing, CNC machining, Vacuum Casting, Injection Molding and more manufacturing services, which enable us to become a one-stop station from prototyping to on-demand production.

In order to meet different market needs, we expand our offerings to service Aerospace, Automotive, Education, Electricals, Jigs & Fixtures, Medical, Oil & Gas, Transportation.

www.in3dtec.com; info@in3dtec.com; +86-21-5059 9658