The Smart Guide to Designing for Manufacturability

A resource for product designers and engineers



Not all mold makers were created equal. Follow the guide of the most advanced.



24541 Maplehurst, Clinton Twp. MI 48036 | www.xcentricmold.com | 586-598-4636

TABLE OF CONTENTS

Injection Molding Basics

Plastic Injection Molding4
Mold Basics5
Advanced Mold Making6
Protoyping7
Materials8-9

Best Practices

Wall Thickness	11
Draft	12
Gates & Runners	13
Tight Tolerances	14
Ribs	15
Bosses	16
Undercuts	17
Corners & Transitions	18

Features to Incorporate

Text on Parts	.20
Hinges & Snaps	. 21
Threading	.22
Overmolding	.23
Insert Molding	.24
Surface Finishes	.25

Avoiding Pitfalls

Knit Lines	27
Sink & Warp	28
Shrink	29

Advanced Molding

Optimizing your part design from concept through production will help you eliminate unneeded costs and reduce the time frame to produce your custom injected molded plastic parts.

Partnering with Xcentric Mold & Engineering will give you the confidence that your parts are in good hands.

Because not all mold makers are the same, this workbook will help you to understand good part design and how you can take advantage of **our advanced**, **proprietary** injection molding process.

We are the only company in the world that can provide **24 hour, interactive quotes** and <u>all</u> of the following in **as fast as 15 days** or less:

- Insert Molding
- Overmolding
- Tight Tolerances
- Internal & External Threads
- Custom Colors
- Unlimited Undercuts
- Unlimited Part Quantities
- Multi-Component Assemblies
- Simple or Complex Parts

- No Design Concessions
- Own your Mold
- Lifetime Mold Guarantee
- FDA & Engineering Grade Resins
- In-house Tooling & Production
- Worry Free Engineering Changes
- Manufacturability Analysis
- Quality Checks

We've defined the standards for quick-turn, injection molding.

If your current supplier can't offer you all of this, you may want to reconsider why they are your current supplier.



Injection Molding Basics

Let's get started by covering the basics of Molds and the Plastic Injection Molding Process.



Plastic Injection Molding

To better understand how to design better parts, learning the injection process is essential. Below is a typical injection molding machine.

The Process:

Plastic *resin pellets* are loaded into the *hopper*, where it travels into the *barrel* of the injection molding machine. Through both heat and pressure, the plastic pellets are melted into a *molten material* that is ready to be injected. As the *screw* turns it creates pressure which will help push the molten plastic through the *nozzle* and into the mold.

With our scientific approach to injection molding, we utilize our highly advanced, **proprietary process engine** to continually monitor and adjust to ensure that each injection molded part is made to exacting specifications consistently. We can minimize warping, size variations and cracking to deliver quality, consistent, stress-free parts every time. Our proprietary process engine creates ideal pressure, temperature and time cycle which is critical to creating high quality custom parts.

Once the right environment inside the barrel is met, the *ram* moves forward driving the screw and channelling the molten plastic into the mold cavity through the nozzle.

Once allowed to cool, the mold opens and the ejection plate engages, releasing the final part from the mold.



Mold Basics

Injection Molds consist of two main components: the mold cavity and the mold core.

CORE (Mold Half B): forms the main internal surfaces of your custom part.

CAVITY (Mold Half A): forms the major external features.

The cavity and core separate (Draw) along the parting line and, with the aid of ejector pins, releases the finished plastic part where the process can be repeated.

Depending on your part design, the parting line can either fall on the top, bottom, stepped or angled in order to accommodate all irregular part features. High quality, efficient tooling relies heavily on good part design as well as advanced skills in mold design and the manufacturing of the tool. An injection mold is a highly precision tool that must be rugged enough to withstand hundreds of thousands of high pressure molding cycles.

By optimizing your part design and focusing on consolidating many key features, you can reduce your overall investment costs significantly.

Xcentric Mold designs and builds the highest quality plastic injection tooling using only the latest in state of the art CAD/CAM and CNC Equipment ensuring you receive your precision injection molded parts fast.

Core (Mold Half B):

Forms the internal surfaces of your custom part, containing the ejection components and often referred to as the noncosmetic side.



Cavity (Mold Half A):

parts often referred to

as the cosmetic side.

Aligning Pins

Forms the external detail for your custom

Draw

Ejector Pins

Advanced Mold Making System

You just learned the basics of molds. But we take mold making a step further with our Advanced Mold Making System which is made up of:

- A Highly Skilled Team
- State of the Art Equipment
- Our Proprietary Software

Together, it allows us to *quickly and cost* effectively convert your CAD drawing into a high quality injection mold *faster* than anyone in the industry.



We are the most advanced mold maker, creating the most simple to the most complex tooling for any of your custom parts.

> Our long lasting tools will produce high quality parts and is backed by our Lifetime Mold Guarantee. You will receive unlimited parts for the life of your project and will **never** be charged for mold repair or rebuild.

In-House tooling enables us to create, modify or change (on-the-fly) your tooling to address engineering changes, reducing downtime and getting your parts in your hands as fast as possible.

We have perfected this system and have been

manufacturing, high quality, precision molds for over 20 years.





Prototyping

With our in-house developers, we have developed software to significantly increase the speed of the mold building process. We can reduce a typical mold build time of 6-18 weeks to just a few days. With this advancement, our customers are able to get real injection molded parts to use either as production or prototype.

Producing prototype parts quickly will help you to get your products to market faster than your competition.

The cost is often less than what most injection molding companies charge for a prototype mold without the life expectancy limitations. Our customers say that they often skip rapid prototyping because our process is so fast and affordable. We can provide production-ready prototype parts as fast or faster than conventional prototyping.

Using our Injection Molding service will help to speed production and reduce your initial cost of production tooling by quickly producing low-volume or prototype parts. By using engineering grade resins, your injection molded prototype parts can be tested under the same conditions as your final parts and can be made of similar, if not the exact, finish materials. This allows you to test in real mechanical, chemical and environmental circumstances and help you create the best possible part design for your product.

With our in-house, high speed mold making, we can also provide Bridge Tooling when you need prototypes using real materials or when your project is not quite ready for full production. Essentially bridging the gap between Prototype and high volume production.

Knowing that your Bridge Tool can be used for production parts, backed by our Lifetime Mold Guarantee, can help save you time and money and will give you the advantage over your competition.



Materials / Resins

High Performance

300°F +

Polyetheretherketone (PEEK) Polyamidimide (PAI) Polyimide Polyphenylene Sulfide (PPS) Polytherimide Polyphenylene Sulfone (PPSU) Polysulfone (PSU)

Engineering Grade

185° - 300°F

185°F

Acetal Nylon Polyesters Polycarbonate Polyurethane Polyphenylene (PPE) Polyvinylidene (PVDF)

Standard Resins

Polypropylene Polyethylene ABS Plastics

- Use standard colors, which are generally less expensive than custom colors
- We can source the material you need.

Material selection will be one of the first and most important steps of designing your part.

So, before you begin, you must consider your parts end function and what properties are required to ensure that the performance and cost of material is optimal.

Producing high quality, consistent plastic injection molded parts relies heavily on your chosen material.



Visit us at: <u>http://www.xcentricmold.com/plastics.php</u> to view detailed information on some of the most common resins to help with your selection. Or give us a call at (586) 598-4636 and speak to one of our knowledgeable sales engineers

However, there are currently 62,000+ thermoplastic resins to choose from and are available in a wide assortment of grades with different properties. For that reason, we recommend you visit <u>http://www.matweb.com</u> where you can browse by name, type or performance characteristics to find the resin you need.

Keep in mind, resins can be combined or added to ensure your finished parts meet your products requirements.

Examples of Additives to Consider:

- Glass Fibers Strengthen/Stiffen resin but can become brittle
- Carbon Fiber Strengthen/Stiffen and static dissipation
- Minerals Increase Hardness
- PTFE Lubrication

- Kevlar Strengthen/Stiffen with less abrasion than glass
- Glass Beads Stiffen and reduce warp
- Stainless Steel Fibers Conductive for electronics
- UV inhibitor Protection from sun



Materials / Resins

Use the chart below to help optimize performance and cost for your chosen material.

	Mech	anical Prop	erties		Mol	dability Prop	perties		
	Low	Average	High		Poor	Average	Good Gre	eat	
		$\overline{}$			•				
Material	Strength	Hi Temp Strength	Impact Resistance	Dimensional Accuracy	Finite Details	Thick Section Voids	Resistance to Sink	Resistance To Flash	Relative Cost
Acrylic				$\mathbf{\bigcirc}$	3			•	\$\$
ABS Plastic				$\mathbf{\bigcirc}$	3	\mathbf{O}		•	\$
Acetal				\mathbf{O}	3	8	\mathbf{O}	•	\$\$
Thermo-Elastomer				8			\mathbf{O}	8	\$\$
High Density Polyethylene (HDPE)				\mathbf{O}			8	8	\$
Nylon 6/6				\mathbf{O}		$\mathbf{\mathbf{O}}$	\mathbf{O}	8	\$\$
Nylon 6/6 (glass-filled)				8	G		$\mathbf{\mathbf{O}}$	$\mathbf{\bigcirc}$	\$\$
Polybutylene (PB)				\mathbf{O}	G		\mathbf{O}	$\mathbf{\bigcirc}$	\$\$\$
Polycarbonate(PC)				\mathbf{O}	3			3	\$\$\$
Polybutylene and Polyethylene				8		$\mathbf{\hat{o}}$	$\mathbf{\hat{o}}$	$\mathbf{\bigcirc}$	\$\$\$
Polypropylene						8	8	8	\$
Polystyrene				\mathbf{O}	\mathbf{O}	-		•	\$



Best Practices

Now that you have the basics covered, let's go over some of the most common best practices for designing your part for the plastic injection molding process.





Wall	Thickness
Wall	Thickness

Material	Recommended Wall Thickness
ABS Plastic	0.045 - 0.140
Acetal	0.030 - 0.120
Acrylic	0.025 - 0.500
Liquid Crystal Polymer	0.030 - 0.120
Long-fiber Reinforced Plastic	0.075 - 1.00
Nylon	0.030 - 0.115
Polycarbonate(PC)	0.040 - 0.150
Polyester	0.025 - 0.125
Polyethylene	0.030 - 0.200
Polyethylene Sulfide	0.020 - 0.180
Polypropylene	0.025 - 0.150
Polystyrene	0.035 - 0.150
Polyurethane	0.080 - 0.750

Next to resin selection, maintaining uniform wall thickness throughout your design is **critical**.

This will help you develop stronger, better looking parts while also aiding in the overall look and feel by reducing blemishes that may occur from stresses during the injection molding process.

Wall thickness will often determine the mechanical performance, cosmetic appearance, moldability and cost-effectiveness of your plastic injection molded custom parts.

Achieving optimal wall thickness is a balance between strength and weight which represents both durability and overall cost. During your design phase, give careful consideration to wall thickness in order to minimize expensive tooling changes down the road.

Utilizing ribs, curves and corrugations can help you to reduce material costs and still provide rigid strength and durability in you plastic molded parts.

- A 10% increase in wall thickness provides approximately a 33% increase in stiffness with most materials.
- Core out unneeded thickness and wall stock
- Use ribs, stiffening features and supports to provide equivalent stiffness with less wall thickness.

Pitfalls of not maintaining uniform wall thickness

Sink & Warp - pg. 28

Shrink - pg. 29



© Xcentric Mold & Engineering. All rights reserved

Draft

Draft is an angle incorporated into your part design to aid in the ejection process from the mold machine.

Plastic parts should be designed with draft to prevent sticking and ejector pin push marks on the show surface during the molding process.

Angles or tapers that you incorporate to key features of your parts such as ribs, walls, posts and bosses that lie perpendicular to the parting line of your part will help to ease the ejection process and eliminate problems when running your injection molded plastic parts.

Less draft will sometimes lead to damaging of your parts during ejection. Also, with plastic molded parts with little or no draft, a mold release agent may have to be used which can cause unwanted reactions and blemishes and may produce additional costs to your finished plastic parts.

A draft angle of 0.5° is the minimum draft needed for most applications. Draft angles of 1.5° to 2° per side are standard for plastic injection molding.

For surfaces that will be textured, a 3° - 5° draft angle is required.





Runners & Gates

Runners and Gates must be designed and incorporated into a mold to ensure that a consistent flow of material fills the mold at the right pressure.

A Gate is the connection between the runner and the molded part. The location and size of the gate is integral to the molding process.

Runners and gates control the flow of the molten material through the mold and into the cavity to create your final plastic part.

Sprue: The main channel in which molten resin enters the mold. This channel is typically larger ensuring that enough material is able to enter the cavity to fill the cavity completely.

Runner System: The runner system connects the sprue to the gate.

Gates: At the opposite end of the sprue, the gates are applied to the runner controlling pressure and flow of molten material. We utilize several gate options to ensure that your part can be filled as completely and consistently as possible.

Gate Location - the location of your gate has a direct impact on moldability. The best positioning is often a balance between ease of molding and part performance.

Gate Scar - Gates can leave minor blemishes so it is important to gate into a non-cosmetic area or where it will not affect part function.

Xcentric will help you design an ideal gate and runner system for your part.



Tight Tolerances

Our customers count on us for expert advice.

With every injection molder you will hear the term "Tight Tolerance".

Unfortunately, that term is thrown about loosely. If not performed correctly, a tight tolerance part can lead to loss in performance or even part failure.

Knowing how to safely and effectively reach our customers exact specifications takes a skilled team and advanced manufacturing technologies.

Generally speaking tight tolerance parts for injection molding is + /- 0.002 inches.

Many factors are put into play such as materials, part complexity, tooling and of course the injection process. Starting with a good part design will ensure tight tolerance repeatability, improved manufacturability and



reduced costs of your plastic injection molded parts.

Size, Geometry and Wall thickness requirements have an impact on tolerance. Thicker walls produce different shrink rates depending on the material, making repeatability difficult.

Before manufacturing, address and analyze your parts making sure to receive both a mold flow analysis and Design for Manufacturability review will help ensure a successful injection molding process and reduce costly delays.

- Utilize low-shrinkage materials for parts with tight tolerances (see page 29).
- Avoiding tight tolerance areas around the alignment of the mold halves (parting line) or moving mold components such as sliders.
- Design your parts to avoid tight tolerance in areas prone to warpage or distortion.



Ribs

Rib Thickness as a percentage of wall thickness

Resin	Minimal Sink	Slight Sink
PC	50%(40% if high gloss)	66%
ABS	40%	60%
PC/ABS	50%	66%
Polyamide (Unfilled)	30%	40%
Polyamide (Glass-Filled)	33%	50%
PBT Polyester (Unfilled)	30%	40%
PBT Polyester (Filled)	33%	50%

Tall Ribs

3 x 0.5T

0 5Т

Often used for structure reinforcement, ribs allow greater strength and stiffness in molded plastic parts without the need to increase the wall thickness. Thicker ribs will often cause sink (see page 28) and other cosmetic problems on the opposite side surface to which they are attached.

As a general rule, design ribs that are approximately 60% of the joining wall thickness for minimum risk for sink marks. Glossy materials, however, require a thinner rib (40% of wall thickness). Keep in mind thin ribs may be more difficult to fill.

Proper rib design involves five parameters:

- **Thickness** (see chart at left). Thickness affects cooling rate and degree of shrinkage which may cause warp.
- Height Should not exceed 3x the rib-base thickness.
- Location Ribs added to uncritical areas can actually reduce impact resistance.

Multiple Ribs

2T

1° Draft

- **Quantity** It's easier to add ribs than remove them so they should be used sparingly and added as needed.
- **Moldability** Thin ribs can be difficult to fill. To be sure, get a manufacturability analysis.

Т

Xcentric will provide you a free Manufacturability Analysis with every quote to ensure your ribs will fill.

> You can replace large, problematic ribs with multiple shorter ribs to provide better performance.





15

Generally,

taller ribs

can cause

issues.

moldability

provide greater support. If not

sized properly, however, it



Bosses are used for locating, mounting and assembly.

Following the guidelines for boss design will have an impact on your final part. Wall thickness and Height are the biggest factors.

Wall Thickness

The wall thickness around a boss design feature should be 60% of the nominal part thickness, if that thickness is less than 1/8". If the nominal part thickness is greater than 1/8" the boss wall thickness should be 40% of the nominal wall.

Height

The height of the boss will also have a role. As a general rule, the height of the boss should be no more than 2-1/2 times the diameter of the hole in the boss.



Undercuts

An undercut is any indentation or protrusion that prohibits an ejection of a part from a one-piece mold. Most commonly categorized by either an internal undercut or external undercut and requires an extra part to capture the detail as part of the mold.

Undercuts typically lead to increased mold complexity and can lead to higher mold construction costs. Usually, a simple re-design of your parts to eliminate or minimize undercuts can lead to lower cost tooling and a more efficient molding process.

When an undercut feature cannot be removed from the part design, it will most likely require internal mold mechanisms to help facilitate the ejection. Typically, the mechanisms consist of side-action slides, jiggler pins, lifter rails, collapsible cores and unscrewing mechanisms.

- When possible, design your part to eliminate undercuts as they will add cost to the mold.
- Other mold makers are not advanced enough and will put limits on undercuts.

Xcentric can provide unlimited undercuts to handle your most complex part.





Corners & Transitions

Corners

Sharp corners can cause molded-in stress from resin flow. It is important to minimize this stress by using rounded corners which also helps to maintain consistent wall thickness. Make the outside radius one wallthickness larger than the inside radius to maintain constant wall thickness through corners.

Transitions

Sometimes it's necessary to transition from thicker walls to thinner ones. Again, sharp corners cause molded-in stress from resin flow. Round or taper the thickness of your transitions to minimize moldedin stresses and stress concentration associated with abrupt changes in thickness.



Features to Incorporate

The following features can all be incorporated into your design to maximize your time and savings.

All in as quick as 1-15 days.





Text on Parts

An added benefit to injection molding parts is the ease of incorporating logo's, labels, instructions or diagrams right onto your custom parts; eliminating secondary costs often incurred with labeling and ensuring clear and precise identification of your custom plastic parts. Whatever the reason, incorporating text onto your plastic parts requires careful consideration and close attention to the details.

Text is often easier to incorporate if it is raised rather than recessed into your part design. Use clear bold letters typically 20 or higher point size for readability and ease of milling. A standard height for raised lettering is just 0.02 inches so do not feel you have to raise your lettering to help it stand out.

Keep your font selection simple and try to avoid serif fonts. Serif fonts tend to incorporate curls or squiggle to the ends of the letters making them difficult to mill.



- Keep Your Text Simple, using thick non-serif fonts.
- 20 point or larger text.
- Utilize raised lettering if possible.



Hinges and Snap Features

Thorough part design can often help to reduced expenses when you face the need for fastening your plastic parts or require additional hardware installation such as hinges or fastening mechanisms. Hinges and snap-fit joints can be incorporated into your plastic parts and reduce or eliminate the need for traditional fasteners such as screws, nuts, washers and spacers.

A part designed with molded-in hinges can replace metal ones while still performing the same function and reducing your products overall cost. When you reduce required hardware, you can lessen the material and assembly cost while also simplifying your design.

Snap joints should be considered during the development of your custom plastic components that need to be secured to other components. Versatile and cost-effective, snap joints and hinges often reduce the cost of secondary hardware expenses and the labor of final assembly.

Polypropylene is the ideal plastic material for integral, injection molded hinges.

Using a hinge to connect the box and cover, allows both parts to be produced in one molding operation. This reduces cost while enhancing functionality.

The hinge must be .060 inch in width and at least .005 inch thick to avoid a sharp bending of the hinge.



Threads

The molding process can incorporate threads right into your custom parts. This eliminates secondary thread cutting that can add unneeded costs to your final part. However, keep in mind thread locations can play a significant role in reducing your total tooling cost.

Placing external threads on the parting line is the most cost effective and easily implemented but can also add the potential for flash or mismatched threads. When threads do not lie centered on the parting line side-actions or slides are required to produce the threads and can potentially add to your molding costs.





Overmolding

Overmolding plastic parts can help in wide range of functional and structural uses. Utilizing two separate injection molds, materials can be bonded together through the injection molding process to enhance the cosmetic looks and/or functionality of your finished plastic parts.

A wide range of materials are capable of being overmolded, including both hard and soft plastic resins. When you choose to overmold your parts most often you can reduce your overall investment by reducing added assembly processes and extra material to manufacture your custom parts.

Careful consideration and planning must be completed ideally from the concept phase and into prototyping. Part design, Mold Design and material selection are important when you plan to overmold your custom plastic components.

20 Years of experience, in-house mold making capabilities and in-house engineering, Xcentric has proven itself a leader in high quality overmolded plastic parts and can provide the solution for your projects specific requirements.

Reasons to Overmold



Insert Molding

Xcentric has over 20 years experience as a qualified custom plastic insert molder. Insert molding is the process of injection molding molten thermoplastic around pieces placed in the injection molding cavity resulting in a strong bond between integral pieces of your final part.

Inserts are offered in a wide variety of materials including plastic, metals, ceramic or any other material that can withstand the pressures and temperatures of the injection mold process.

There are many uses for plastic injection insert molding. Placing threads or securing wire connectors, knobs, controls, warnings, labels and electronic devices.

Insert molding is an effective and cost-efficient solution for reducing a products overall cost, by incorporating parts into the molding process which would otherwise require secondary assembly or installation.

Accurate mold design and construction is essential to insert molding to not only maintain part tolerances but also assure the tooling reliability.







Tooling Marks Removed

B3

Smooth Finish

B2

High Polish Finish

A3

Highest Polish Finish

A2

Rough Texture

MT-11010

Rough Texture

MT-11020

Surface Finishes

Xcentric offers a wide range of materials with multiple surface finish options. Most resins are available in many colors and we are also capable of creating custom colors to match your requirements.

If you are working on a project that may require painting as a final process, consider utilizing molded-in color which can often be achieved for a much lower price than traditional painting labor and material costs. If you must paint your plastic parts, select a resin that paints easily and preferably one that does not require surface etching and/or primer.

Xcentric Mold offers a variety of surface finishes to add function as well as cosmetics to your finished plastic injection molded parts. Whether you require a mirror like gloss finish or a textured finish for grip and usability, we have the available solution for your plastic part needs.

Xcentric Surface Finishes Available

- B3 320 Paper Finish (Standard)
- B2 400 Paper (Smooth)
- A3 Polish (High Gloss Mirror Finish)
- A2 Optical Polish (Gloss Finish)
- MT- 11010 Bead Blast (Textured)
- MT- 11020 Bead Blast (More Textured)



Avoiding Pitfalls

Learn what can happen to your parts in the injection molding process if you ignore these guidelines.



Knit Lines

The injection molding process is fairly simple, plastic resin is heated to its melting point and forced through the machine and into your mold to produce your plastic parts.

The leading edge of the molten material is often the coolest point and the closest to solidifying. When the molten plastic meets an obstruction it must travel around and meet at the other side. If the plastic has cooled too much during the injection process it can lead to knit lines in plastic parts when they meet past an obstruction.

ABS is the most common resin to be prone to knit lines.

If you are concerned about potential knit lines, turn to the mold flow analysis of your part and address any design issues that can be easily modified. Review similar materials that may be less prone to show knit lines.

With good part design and a well designed mold, knit lines can often be significantly reduced or removed completely.

Potential for Knit Lines as molten plastic travels around a barrier and begins to cool.



Sink and Warp

Variations of shrinkage in materials can lead to warp, distortion and dimensional issues with injection molded parts.

As the plastic material cools, the molecules that make it up move closer together. If the cooling rate is different, such as thinner or thicker walls the stress caused by cooling can lead to the material wanting to warp.

As the plastic in the mold cools from the outside in, it can cause pulling on the outer walls resulting in sink marks. Thinner wall thickness will help to prevent this. Where possible, always try to design a part with thinner and consistent wall sections to minimize warp and sink marks.

Careful consideration to part and mold design must be addressed in order to create high quality, consistent plastic parts and at Xcentric, we provide you with the tools and experience to reduce or eliminate potential cosmetic or structural defects of your plastic custom parts.

When you partner with Xcentric, you gain the advantage of our Advanced Mold Making System which will help you to create the best quality parts at the lowest price in the industry.



Shrink

Material	Shrink inch / inch
Polypropylene -unfilled	0.015" - 0.018"
Polyethylene	0.020" - 0.025"
ABS	0.0035"
HIPS	0.0035"
GPPS	0.0035"
Polycarbonate	0.007"
PC-ABS	0.007"
Acrylic	0.003" - 0.004"
Nylon 6/6 (PA66) - Unfilled	0.020"
Nylon 6/6, 33% short glass	0.0035"
PBT	0.015"
Acetal (POM)	0.020"
Acetal Homopolymer	0.018" - 0.020"
PVC (Rigid)	0.0035"
TPE (Santoprene)	0.014" - 0.018"
Noryl	0.005" - 0.007"
Noryl 30% glass filled	0.001"
TPU	0.007" - 0.010"
Polysulphone	0.007"

With most injection molding resin a certain degree of shrink can be expected due to the materials chosen. Some materials tend to shrink more than others so careful consideration on material choice should be made.

Rapid changes to wall thickness are the most common cause of shrinkage due to the pressures exerted for the plastic material to fill your mold. When designing your parts try to eliminate thin wall sections leading into thicker wall sections and create parts with uniform wall thickness throughout.

If thick and thin sections are necessary try to transition the change gradually, utilizing angles to help aid the flow of materials throughout your plastic parts.

Controlling part shrinkage is critically important especially in tight tolerance plastic parts.

At Xcentric, our process engines allow us to produce your parts to very tight tolerances by analyzing every aspect of the injection molding process to ensure consistent, stress-free manufacturing of all your custom plastic parts to help you reduce or eliminate manufacturing defects.



Summary

BEST PRACTICES

Resins/Materials

- Use standard colors, which are less expensive than custom colors
- Compare the price of materials that meet your product requirements, but avoid making your selection based upon price alone

Wall Thickness

- Maintain uniform Wall Thickness throughout
- Utilize Ribs to reinforce walls without adding to thickness
- A 10% increase in thickness = 33% increase in stiffness
- Core out unneeded thickness and wall stock

Draft

• Maintain a minimum of 0.5° draft angle on all features perpendicular to the parting line. 1° - 2° is ideal.

Tight Tolerances

• Utilize low-shrinkage materials for parts with tight tolerances

Ribs & Bosses

• Design ribs and bosses to approximately 60% of the joining wall thickness for minimum risk for sink marks.

Undercuts

• Undercuts will add cost to the mold. Minimize them when you can. Otherwise, there are no limits.

Corners and Transitions

- Use gradual transitions if wall thickness must change.
- Corners: R1 + T = R2

Text on Parts

- Use simple, non-serif fonts
- Raised lettering molds better than recessed
- Use 20 point or larger text

Hinges and Snaps

• Used to simplify assembly, enhance function and reduce cost

FEATURES

Threads

- Stop threads short of the end
- Limit thread pitch to 32 treads per inch

Overmolding

- Design holes or slots in the first mold to interlock pieces physically
- Overmold for color contrast, soft grips or to eliminate assembly

Insert Molding

• Use for securing threads or other functional pieces to the part and eliminating assembly

Surface Finishes

• Add aesthetics to your parts and choose from a variety of finishes





Faster Parts. Your Way™

Our low-volume, custom part manufacturing process is faster than anyone in the industry. And we do not impose any design restrictions. The part you designed is the part you get.

In short, if we can't manufacture your most complex part and deliver it in 15 days or less...NO ONE CAN.

20 years in business, Xcentric has been delivering low cost, high quality custom metal and plastic parts for every industry. Utilizing proprietary processes and our advanced mold making department allow us to provide you both simple and complex parts in as little as 1-15 days.

Plastic Injection Molding - 1-25,000 parts in a range of engineering grade resins allows you to quickly test your parts with real materials quickly, getting your parts to market faster than you ever thought possible.

CNC Machining - Precision machines cut your parts from plastic or metal, ensuring you high quality and even faster delivery.

Rapid Prototyping - SLA, SLS, FDM or Cast Urethane Molding, Xcentric mold can deliver your prototype parts quickly allowing you to efficiently fine tune your design and speed the manufacturing process to production.

We have built the company from the ground up, ensuring both our services and people maintain or exceed our high expectations. From manufacturing to customer service, our team will provide you support and guidance throughout your project.

This workbook was developed to help product designers learn more about the process in order to more effectively design custom parts

We hope you find the information useful and we look forward to the opportunity of earning your business.



