XCENTRIC
MOLD & ENGINEERING

GUIDE

THE SMART GUIDE TO DESIGNING FOR MANUFACTURABILITY

A design resource for plastic injection molding.

CONTENTS

ADVANCED MOLDING

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

In this eBook you will find:

- Best practices for designing a part for injection molding.
- Tips on features you may incorporate into your design.
- Guidance on avoiding common pitfalls.

Whether you are in the design stage or ready for production, our manufacturing experts can help you with your project. From Design to Prototyping, Pre-Production to Production, our team is ready to support you.

INJECTION MOLDING BASICS

The basics of molds and the plastic injection molding process.

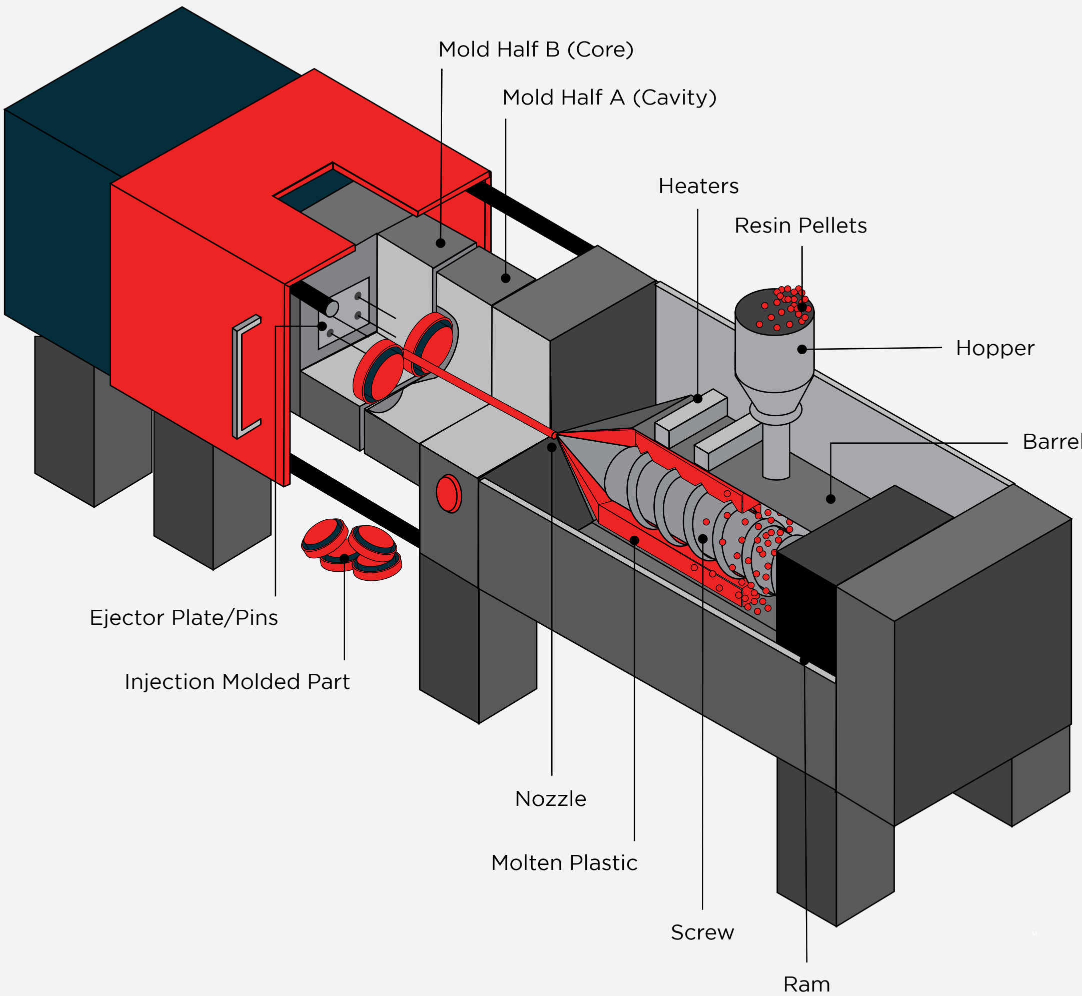
PLASTIC INJECTION MOLDING

To understand part design, it is essential to learn the plastic injection molding process. The illustration depicts a typical injection molding machine.

THE PROCESS: Plastic resin pellets are loaded into the hopper. The pellets then travel 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. Pressure, temperature, and time cycle are optimized to create quality custom parts.

Once the right environment inside the barrel is met, the ram moves forward driving the screw. As the screw turns, it creates pressure, which pushes the molten plastic through the nozzle and into the mold.

Once cooled, the mold opens and the ejector 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.

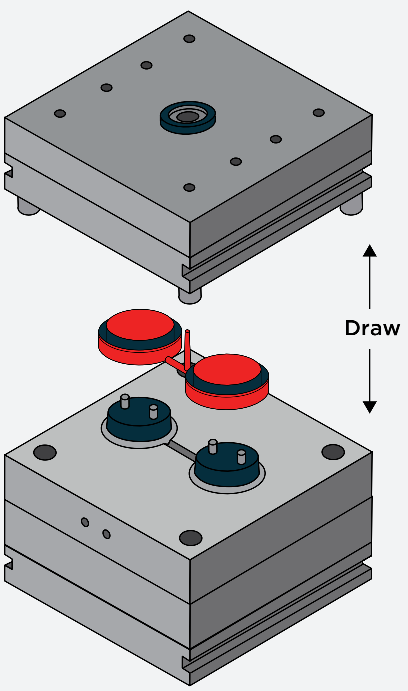
CAVITY (MOLD HALF A): forms the major external features.

CORE (MOLD HALF B): forms the main internal surfaces of the part.

The cavity and core separate (Draw) along the parting line and, with the aid of ejector pins, release the finished plastic part. The process is then repeated. Depending on your part design, the parting line can either fall on the top, bottom, stepped or angled in order to accommodate all 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 high 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

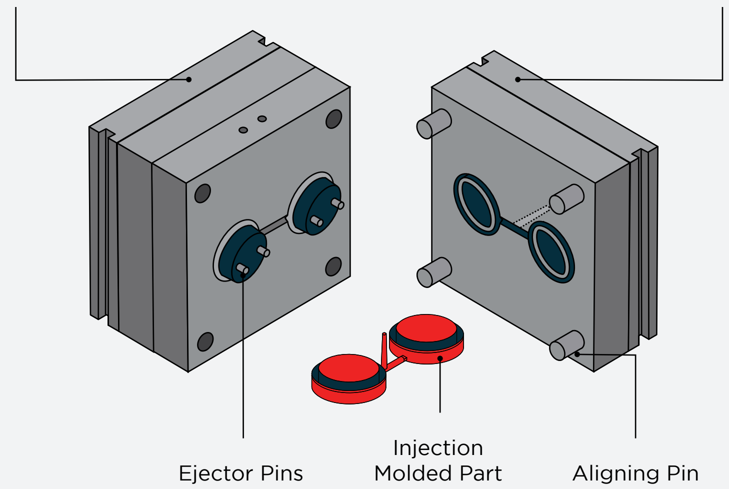


CORE (MOLD HALF B):

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

CAVITY (MOLD HALF A):

Forms the external detail for your custom parts and often referred to as the cosmetic side.



TYPES OF MOLDS

When you approach a custom injection molder to produce complex plastic parts or components, they will first help you determine if the product is suitable for injection molding. Experienced molders will ask a series of questions to better understand your project needs like design, purpose, and objectives, in order to:

- Determine the overall moldability of your part.
- Confirm what type of mold is best suited for the job.

Standard molds are defined by the Plastics Industry Association (PIA). These mold standards are categorized into five classifications to guide quotes and orders based on uniform mold types.

5 TYPES OF MOLD CLASSIFICATIONS

	CYCLES	PRODUCTION LEVEL	USES	INVESTMENT
1 Class 101 Mold	1 million or more	**Extremely high**	Extremely high production and fast cycle times.	Class 101 molds are the highest priced and made with only the highest quality materials.
2 Class 102 Mold	Not exceeding **1 million**	**Medium to high**	Good for parts with abrasive materials and/or tight tolerances.	Class 102 molds are high-priced and made with materials of high quality.
3 Class 103 Mold	Not exceeding **500,000**	**Medium**	A very popular mold for low to medium production parts.	Class 103 molds fall within common price ranges.
4 Class 104 Mold	Not exceeding **100,000**	**Low**	Good for limited-production parts with non-abrasive materials.	Class 104 molds fall within low to moderate price ranges.
5 Class 105 Mold	Not exceeding **500**	**Very low**	Prototype only.	Class 105 molds are built inexpensively to produce a very limited number of product prototypes.

Simply reviewing a mold classification chart isn't enough since each mold classification has unique parameters. It's essential that you partner with a molder that is knowledgeable about these mold standards and the classification spectrum.

The molder's guidance can help you determine the best type of mold that meets your project, cost, and quality objectives.

MOLDING OPTIONS TAILORED TO YOUR NEEDS

Using mold classifications to determine the correct mold type is crucial to achieving quality, production, and cost objectives. At Xcentric, we provide prototype injection molding and low-volume injection molding services.

Our streamlined prototype injection molding service can deliver first articles in as fast as five days. Meanwhile, our low-volume production process frequently—and rapidly—delivers plastic parts for customers needing a few hundred to thousands of components as a bridge to higher productivity molds.

These are the types of molds available:

	PROTOTYPE	LOW-VOLUME PRODUCTION
Objective	• Design validation • Forms, fits, and function parts • Materials trials	• Low-volume production parts • Bridging the gap from prototype to production
Ideally When...	• Lifetime volumes are below 2,500 parts • An affordable entry point for tooling is important • Part production is only needed for 1 year or less	• Part production is needed over multiple years • Lifetime volumes exceed 2,500 parts • Lower part price is critical
Mold Costs	• Lower	• Higher
Part Costs	• Higher	• Lower
Timing	• 3-4 weeks	• 3-5 weeks
Mold Cavities	• Single cavity	• Single & multi-cavity
Materials	• Standard Xcentric materials or customer-provided	• Standard Xcentric materials, custom, or customer-provided
Finishes	• XME-standard basic blasts & polishes	• XME-standard basic blasts, polishes, custom
Mold Guarantee	• Limited to 2,500 shots	• Lifetime/unlimited
Mold Ownership	• Yes	• Yes
Mold Storage	• 24 months of inactivity	• 36 months of inactivity
Quality Documentation	• Basic part inspection upon request	• Basic part inspection included • Additional inspection services available
Base Tool Material	• High-grade QC-10 aluminum	• High-grade QC-10 aluminum
Set Up Fees	• Yes	• Waived on orders of 10,000 pieces or more

TIPS ON HOW TO CHOOSE YOUR MOLD TYPE

1 **CHOOSE PROTOTYPE MOLDS** if speed to market is a priority. This means you require a short build time and will need to make quick modifications should the injection molding process or the part require them.

---

2 **CHOOSE PROTOTYPE MOLDS** if your part needs to be tested in real mechanical, chemical, and environmental circumstances. Injection molded prototype parts made with engineering-grade resins can be tested under the same conditions as your final parts, and can be made of similar, if not the exact, finish materials.

---

3 **CHOOSE PROTOTYPE MOLDS** if you require a low volume of production. When designed and built correctly, prototype tools can be used to bridge the gap between prototype and production.

Using prototype tools for bridge production enables companies to release production parts into the marketplace quicker than if they waited for production tooling to be built, thereby accelerating revenue attainment, and giving them an advantage over their competition. For low-volume production, prototype molds are often all that is required.

---

4 **CHOOSE PRODUCTION MOLDS** if you're ready to go into high-volume part production. Typically, traditional molds are made of steel. Costs are higher than prototype molds because production molds must be made of a durable material to endure high-volume part production. Production molds usually take more time to build than prototype molds and are not easily modified. At Xcentric, production molds are made with high-grade QC-10 aluminum that can endure high-volume production and costs less to build.

MATERIAL SELECTION

Material selection is one of the first and most important steps of designing your part. This is because high-quality plastic injection molded parts made to meet your part or product's end-use requirements rely heavily on the material they're made with.

There are thousands of plastic variations available, from standard resins to high-performance polymers. Here are some examples:

HIGH PERFORMANCE 300°F +

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

ENGINEERING GRADE 185°–300°F

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

STANDARD RESINS 185°F

- Polypropylene
- Polyethylene
- ABS Plastics

To select the best one for your needs, it's helpful to narrow down the choices. Review these five tips before your next project to land on the perfect material for your plastic part:

1. Factor in material sourcing, selection, and testing when setting project timelines.
2. Give yourself enough time and resources to vet through all the plastic grades that are available to you.
3. Don't underestimate the volatility of the materials market, which influences availability.
4. Make sure to define your part's application requirements. What's the end-use requirement? Who are the end-users? Under what conditions will it operate? What regulations do you need to meet?
5. Don't neglect aesthetics. It affects material selection, filler composition, and finishing requirements.

> View detailed information here on the more than 40 resins Xcentric has in stock for you to choose from. We also recommend you visit www.matweb.com where you can browse by name, type, or performance characteristics to find the resin you need.

You may choose to purchase materials yourself before approaching a molding partner, but your injection molding partner should also be able to source the material you need. Xcentric can guide you through this process before making a formal material selection.

STEP-BY-STEP GUIDE TO PICKING THE BEST MATERIAL FOR YOUR NEEDS

YOUR DESIGN	Inventor/designer shows Xcentric their design.
DEFINE APPLICATION REQUIREMENTS	Xcentric collaborates with inventor/designer to determine end-use needs.
CONSULTATION WITH MOLDING PARTNER	Xcentric helps them weigh plastic options based on shrinkage and warp. Xcentric can help them adapt material (and source, if required).
MATERIAL SELECTION	Material is formally selected.
MOLD TRIAL AND SAMPLING	Mold cut and trialed, samples tested to identify issues.
PRODUCTION	Material and mold are approved, and production begins.

MATERIALS

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

MATERIAL	STRENGTH	HI TEMP STRENGTH	IMPACT RESISTANCE	DIMENSIONAL ACCURACY	FINITE DETAILS	THICK SECTION VOIDS	RESISTANCE TO SINK	RESISTANCE TO FLASH	RELATIVE COST
Acrylic	Average	Low	Low	Good	Good	Great	Good	Good	$$
ABS Plastic	Average	Low	High	Good	Good	Good	Good	Good	$
Acetal	Average	Low	Average	Good	Good	Poor	Good	Good	$$
Thermo-Elastomer	Low	Low	High	Poor	Great	Great	Good	Poor	$$
High Density Polyethylene (HDPE)	Low	Low	High	Good	Great	—	Poor	Poor	$
Nylon 6/6	Average	Low	High	Good	Great	Good	Good	Poor	$$
Nylon 6/6 (Glass-filled)	High	High	Average	Poor	Good	Great	Good	Good	$$
Polybutylene (PB)	Average	Low	High	Great	Great	Average	Good	Good	$$$
Polycarbonate (PC)	Average	High	High	Great	Great	Average	Good	Good	$$$
Polybutylene and Polyethylene	High	Average	Average	Poor	Good	Good	Good	Good	$$$
Polypropylene	Low	Low	High	Good	Great	Poor	Poor	Poor	$
Polystyrene	Average	Low	Low	Good	Good	—	Good	Good	$

Mechanical Properties			Mechanical Properties			
Low	Average	High	Poor	Average	Good	Great

FILLERS

Resins can be combined, and additives added to them to ensure your finished parts meet your product's requirements.

Most additives function as either fillers or reinforcements:

- **FILLERS**—added to only extend the volume of a polymer.
- **FUNCTIONAL**—fillers are added to a polymer to improve specific characteristics.
- **REINFORCEMENTS**—added to significantly increase the modulus (stress/strain curve) of the polymer.

ADDITIVES

- Impact modifiers
- Colorants
- Blowing agents/foaming agents
- Fillers (extenders)
- Coupling agents
- Lubricants
- Plasticizers
- Flame retardants

FILLERS	REINFORCEMENTS
Talc	Glass fiber
Carbon black	Long glass fiber
Calcium carbonate	Carbon black
Wollastonite	Glass hollow bead
Kaolin clay	Carbon fiber
Silica	Graphite
Barium sulfate	Ethylene propylene diene monomer (EPDM)

EXAMPLES OF ADDITIVES TO CONSIDER

1. **GLASS FIBER**—Strengthens/stiffens resin but can become brittle
2. **CARBON FIBER**—Strengthens/stiffens resin and provides static dissipation
3. **MINERALS**—Increases hardness
4. **PTFE**—Lubrication
5. **KEVLAR**—Strengthens/stiffens resin and is less abrasive than glass
6. **GLASS BEADS**—Stiffens and reduces warp
7. **STAINLESS STEEL FIBERS**—Conductive for electronics
8. **UV INHIBITORS**—Protection from the sun

BEST PRACTICES

Common best practices for designing parts for the plastic injection molding process.

WALL THICKNESS

After resin selection, maintaining uniform wall thickness throughout your design is critical.

Optimizing wall thickness will help you develop stronger and better looking parts while also reducing blemishes and possible part warp.

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, and directly affects both durability and overall cost. During design, give careful consideration to wall thickness in order to minimize expensive tooling changes down the road.

Utilizing ribs, curves and corrugations can help you reduce material costs and still provide rigid strength and durability in your 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—page 32
- Shrink—page 33

MATERIAL	RECOMMENDED WALL THICKNESS (INCHES)
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

DRAFT

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

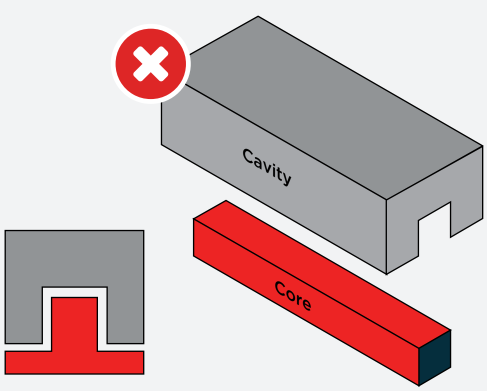
Plastic parts should be designed with draft to prevent sticking and ejector pin push marks on the show surface during the molding process.

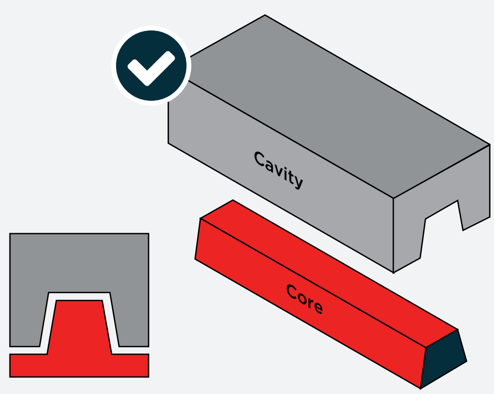
Angles or tapers that you incorporate to key features of your parts such as ribs, walls, posts and bosses that lie perpendicular to the parting line of your part will help ease the ejection process and eliminate problems when running your injection molded plastic parts.

Less draft can sometimes lead to damaging 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 typically 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.

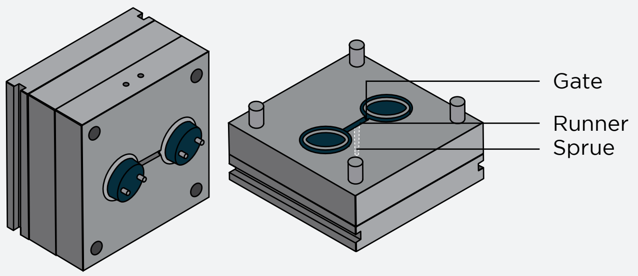
GATES—At the opposite end of the sprue, gates are applied to the runner to control pressure and flow of molten material. Several gate options are used to ensure that a part can be filled as completely and consistently as possible.

GATE TYPES—Edge gates are most common, with fan gates and chisel gates being variations of edge gates. Other gate types include tab, tunnel, pinpoint, filter-bowl and diaphragm gates.

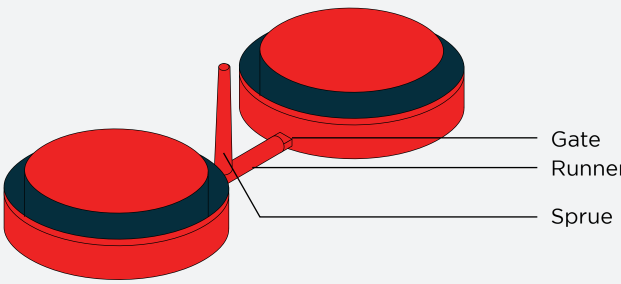
GATE LOCATION—the location of your gate has a direct impact on moldability. The best positioning is often a balance between ease of molding and part performance.

GATE SCAR—Gates can leave blemishes so it is important to gate into a non-cosmetic area and where it will not affect part function.

The Mold



The Part



CRITICAL FEATURES

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

Many factors influence the success of a part including materials, part complexity, tooling and the process itself. 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 to help ensure a successful injection molding process and reduce costly delays.

Utilize low-shrinkage materials for parts with tight tolerances (see page 33).

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 tolerances in areas prone to warp or distortion.

RIBS

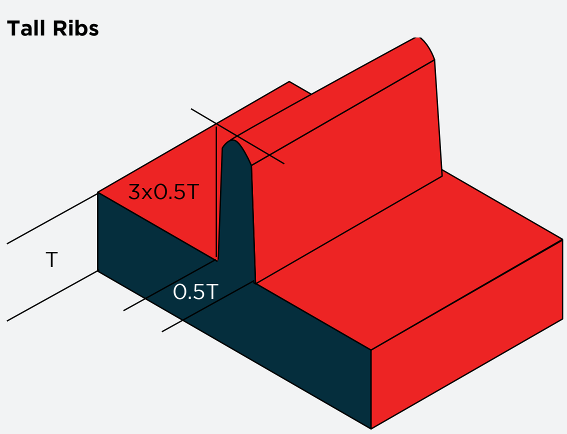
Often used for structural reinforcement, ribs allow greater strength and stiffness in molded plastic parts without the need to increase the wall thickness. Thick ribs will often cause sink (see page 32) and other cosmetic problems on the opposite side surface to which they are attached.

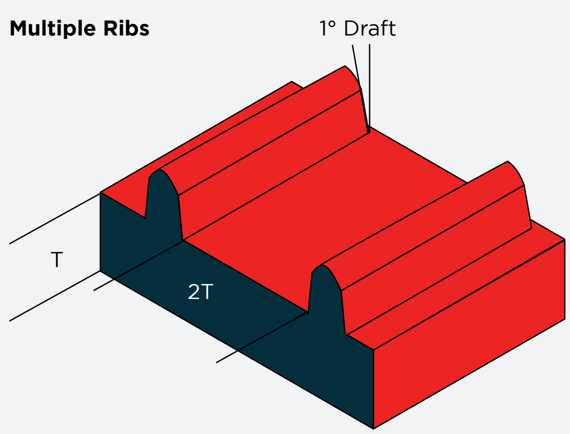
As a general rule, design ribs that are approximately 60% of the joining wall thickness to minimize risk of sink marks. Glossy materials, however, require a thinner rib (40% of wall thickness). Keep in mind thin ribs may be more difficult to fill.

- **THICKNESS**—see chart (right). 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.
- **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. Always get a manufacturability analysis to be sure ribs fill completely.

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%





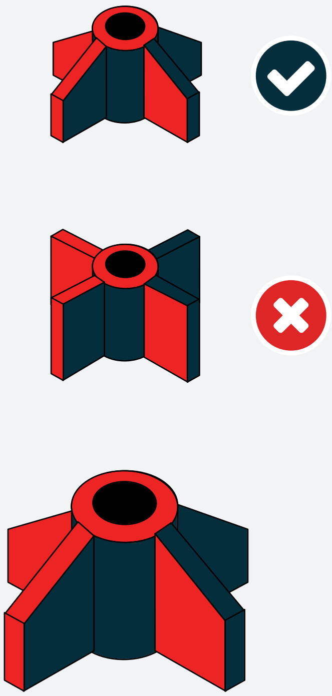
BOSSES

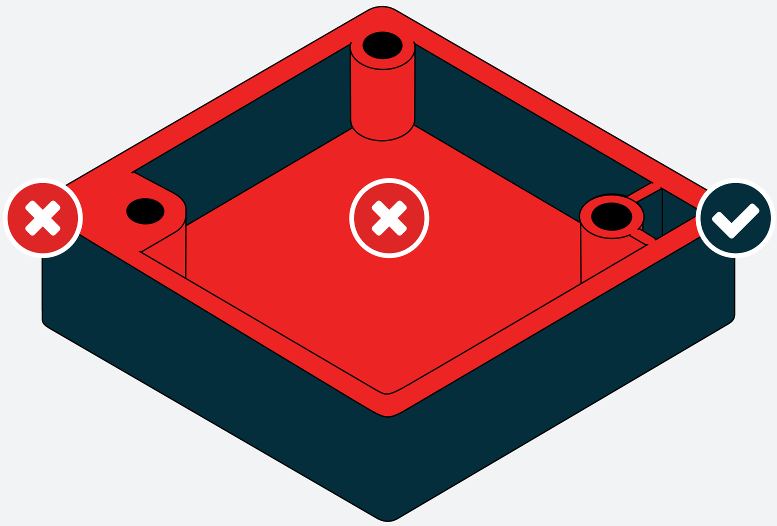
Bosses are used for locating, mounting and assembly.

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.



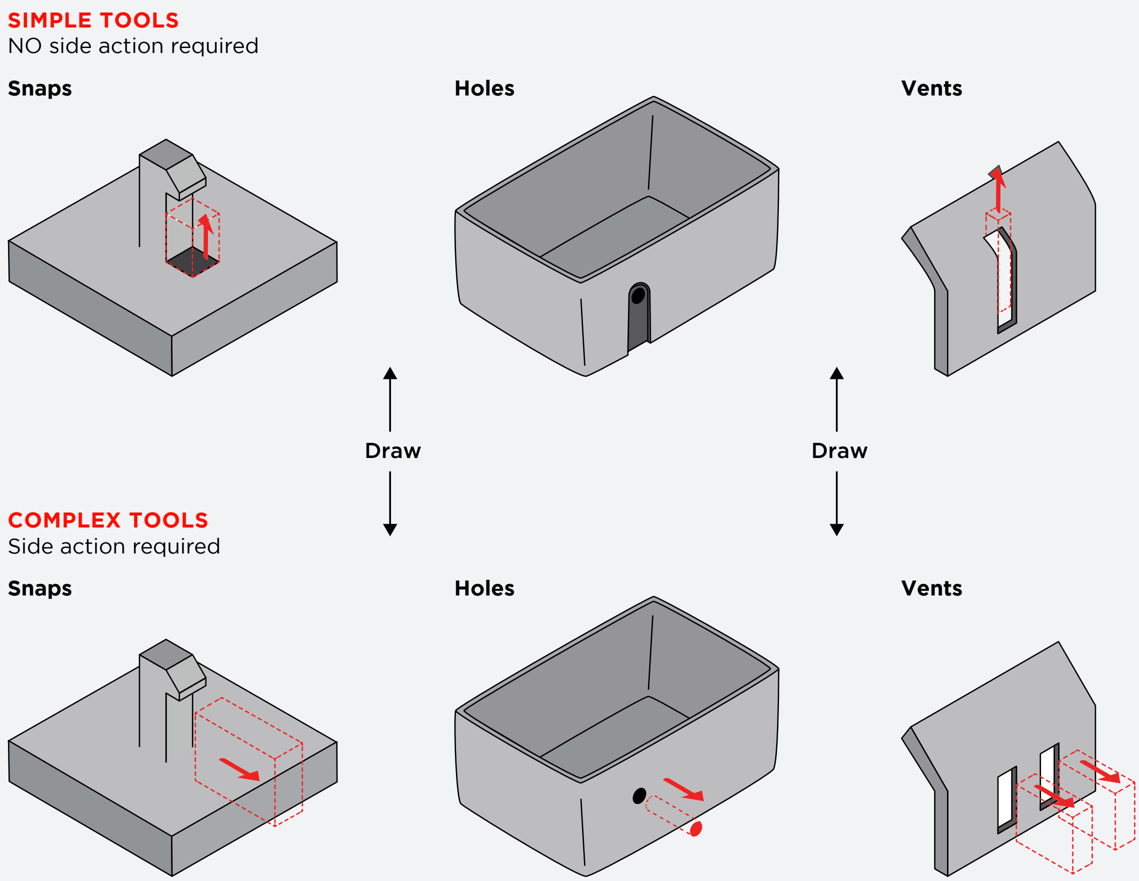


UNDERCUTS

An undercut is any indentation or protrusion that prohibits an ejection of a part from a one-piece mold. These are 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 the part 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.



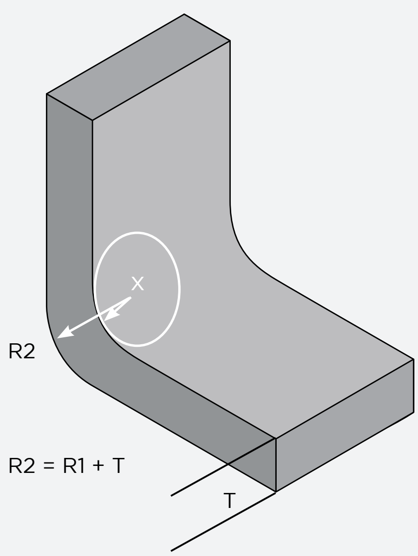
CORNERS & TRANSITIONS

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.

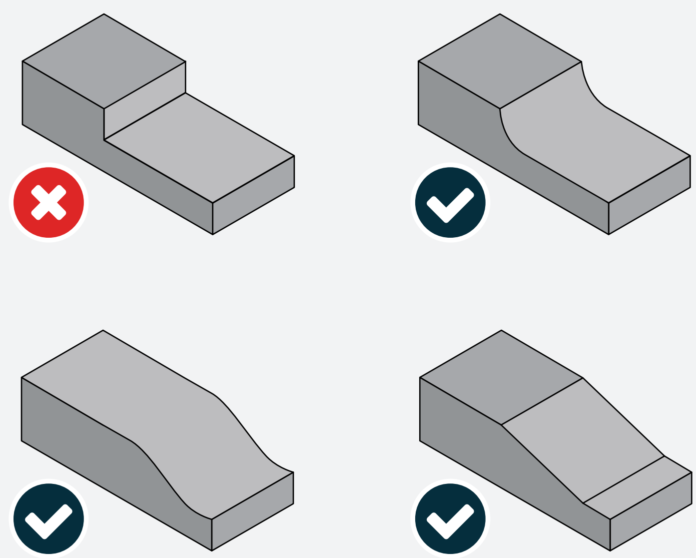
Make the outside radius one wall thickness larger than the inside radius to maintain constant wall thickness through the corners.

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 molded in stresses and stress concentration associated with abrupt changes in thickness.

Corners



Transitions



FEATURES TO INCORPORATE

The following features can be incorporated into your design to enhance your part, potentially saving time and money.

TEXT ON PARTS

An added benefit to injection molded parts is the ease of incorporating logos, labels, instructions, or diagrams right onto your parts.

This can eliminate secondary costs often incurred with labeling and ensure clear and precise identification of your plastic parts.

Whatever the reason, incorporating text onto your plastic parts requires careful consideration and close attention detail.

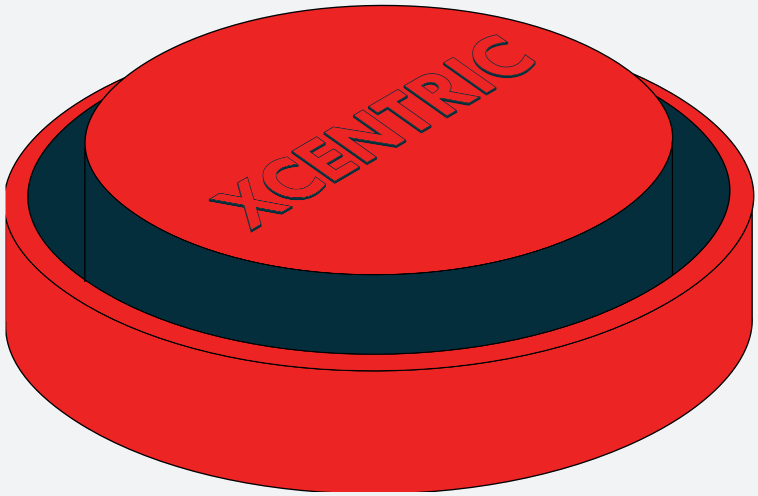
Text is often easier to incorporate if it is raised rather than recessed into your part design. Use clear, bold letters typically 20 or higher in font size for readability and ease of milling. A standard height for raised lettering is 0.02 inches; do not feel you have to raise your lettering to help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

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

Keep your text simple, using thick non-serif fonts.

Size 20 font or larger.

Utilize raised lettering, if possible.



HINGES & SNAP FEATURES

Thorough part design can often help to reduce 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 to reduce or eliminate the need for traditional fasteners such as screws, nuts, washers and spacers.

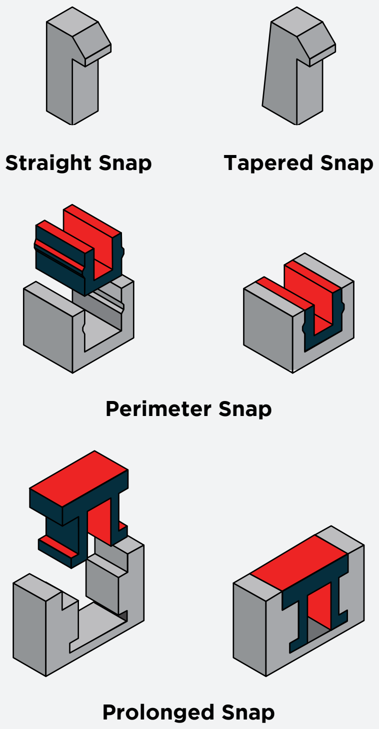
A part designed with molded-in hinges can replace metal ones while still performing the same function and reducing your product's overall cost. When you reduce required hardware, you can lessen the material and assembly cost while also simplifying your design.

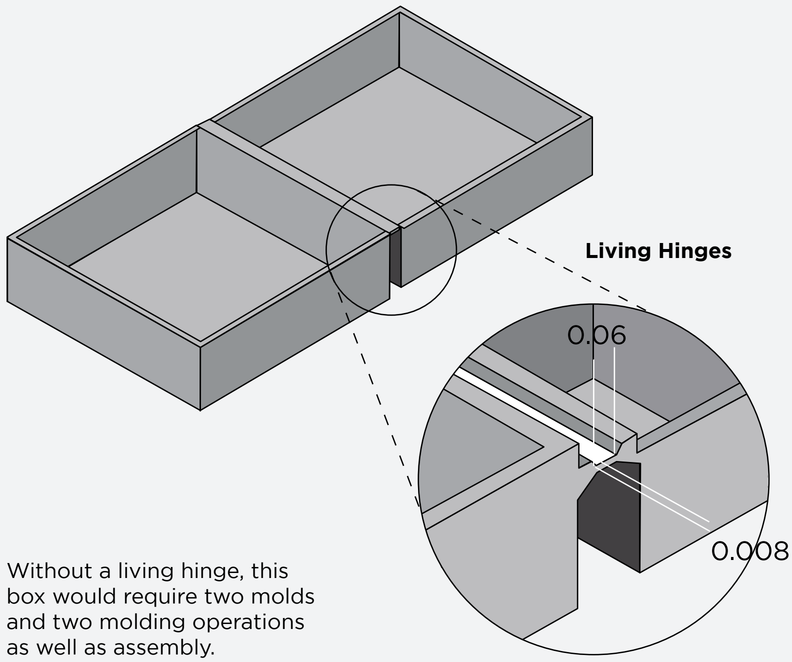
Snap joints should be considered during the development of your custom plastic components that need to be secured to other components. Versatile and cost effective, snap joints and hinges often reduce the cost of secondary hardware expenses and the labor of final assembly.

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 inches in width and at least .008 inches thick to avoid a sharp bending of the hinge.





Without a living hinge, this box would require two molds and two molding operations as well as assembly.

THREADS

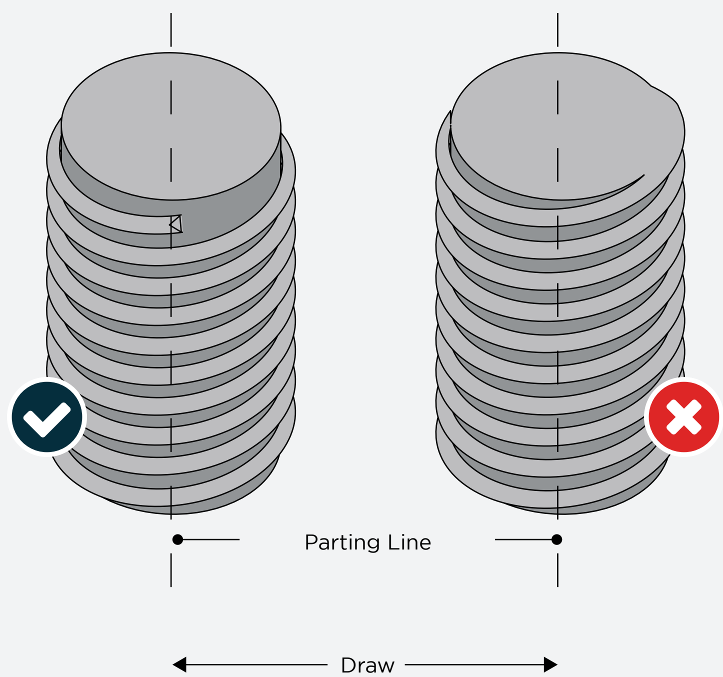
The molding process can incorporate threads right into your custom parts. This eliminates secondary thread cutting that can add unnecessary costs. Keep in mind thread locations can play a significant role in reducing your total tooling cost.

Placing external threads on the parting line is cost effective and easily implemented.

However, it 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. This can potentially add to your molding costs.

> Stop threads short of the end to avoid making thin, feathered threads that can easily cross-thread.

> Limit thread pitch to no more than 32 threads per inch for ease of molding and protection from cross threading.



OVERMOLDING

Overmolding plastic parts can help in a 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 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 you can reduce your overall investment by reducing added assembly processes and extra material required to manufacture your parts.

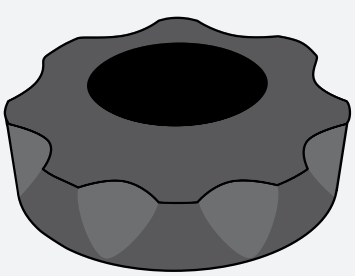
Careful consideration and planning for overmolding must happen in the concept phase.

Part design, mold design and material selection are important when you plan to overmold plastic components.

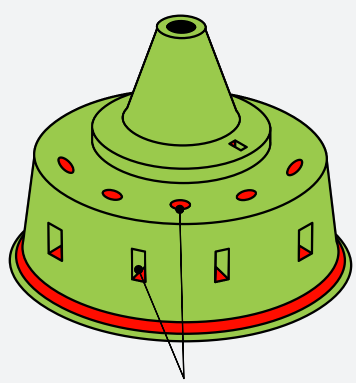
REASONS TO OVERMOLD:

- To add aesthetically pleasing color contrasts.
- To provide a soft grip surface.
- To add flexibility to rigid part areas.
- To eliminate assembly.
- To capture one part inside of another without having to use fasteners or adhesives.

2nd Plastic Injection



1st Plastic Injection



It’s good practice to design features like holes and slots into your overmolded parts to help them interlock.

Finished Part

INSERT MOLDING

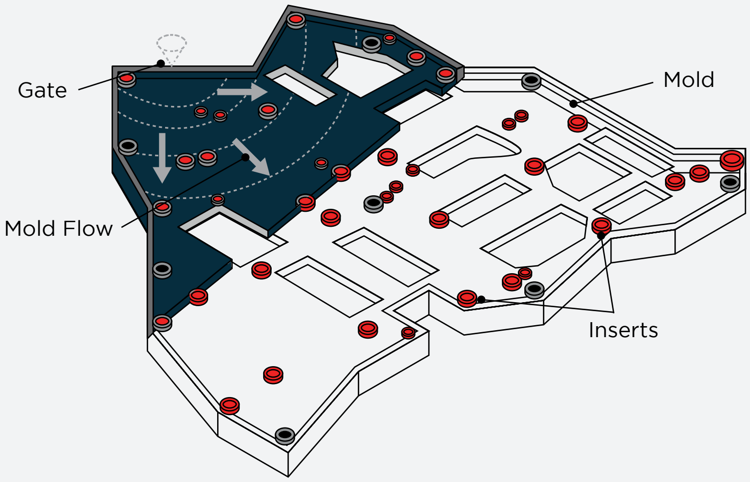
Insert molding is the process of injection molding molten thermoplastic around pieces placed in the injection molding cavity resulting in a strong bond between the integral pieces of your final part.

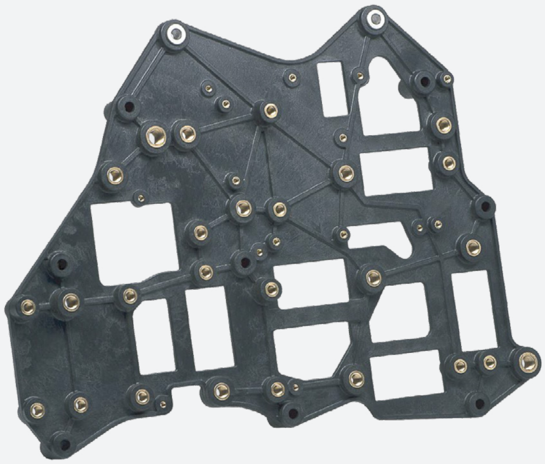
Inserts are offered in a wide variety of materials including plastic, metal, ceramic or any other material that can withstand the pressures and temperatures of the injection mold process.

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

Insert molding is an effective and costefficient 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 not only to maintain part tolerances, but also to assure the tooling reliability.





SURFACE FINISHES

Consider incorporating surface finishes to add function as well as cosmetic appeal to final plastic injection molded parts. Finishes can give a mirror-like gloss finish or a textured finish for grip and usability.

SPI* STANDARD	FINISH	FINISHING METHOD
A-2*	High Glossy Finish	Grade #6, 3000 Grit Diamond Buff
A-3*	Normal Glossy Finish	Grade #15, 1200 Grit Diamond Buff
B-2*	Medium Semi Glossy Finish	400 Grit Paper
B-3*	Normal Semi-Glossy	320 Grit Paper
XT-1	Light Texture	Light Bead Blast
XT-2	Heavy Texture	Heavy Bead Blast

*SPI (Society of Plastic Industry) Standards

Request a free 8-piece surface finish sample kit now.

PAINTING

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 at a 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 a primer.

AVOIDING
PITFALLS

Follow these guidelines to manage the unexpected issues that may occur with your parts during the injection molding process.

KNIT LINES

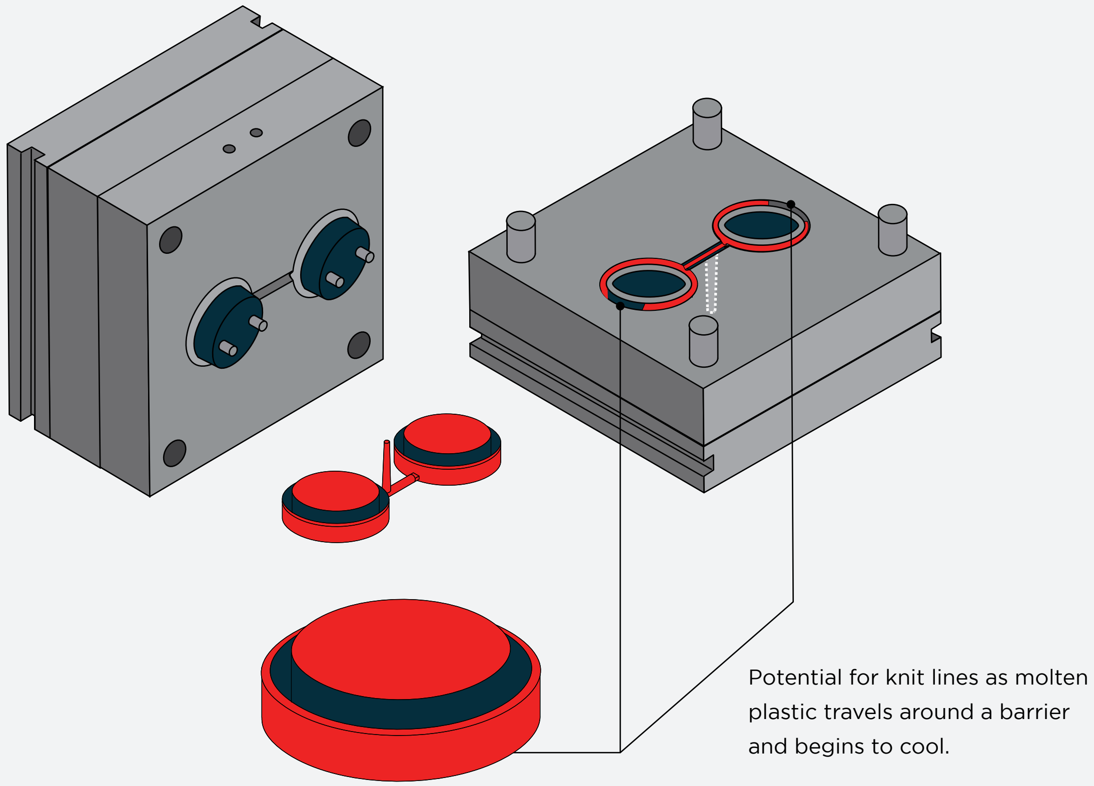
The injection molding process is fairly simple. Plastic resin is heated to its melting point and forced through the machine and into your mold to produce your plastic parts.

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 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.



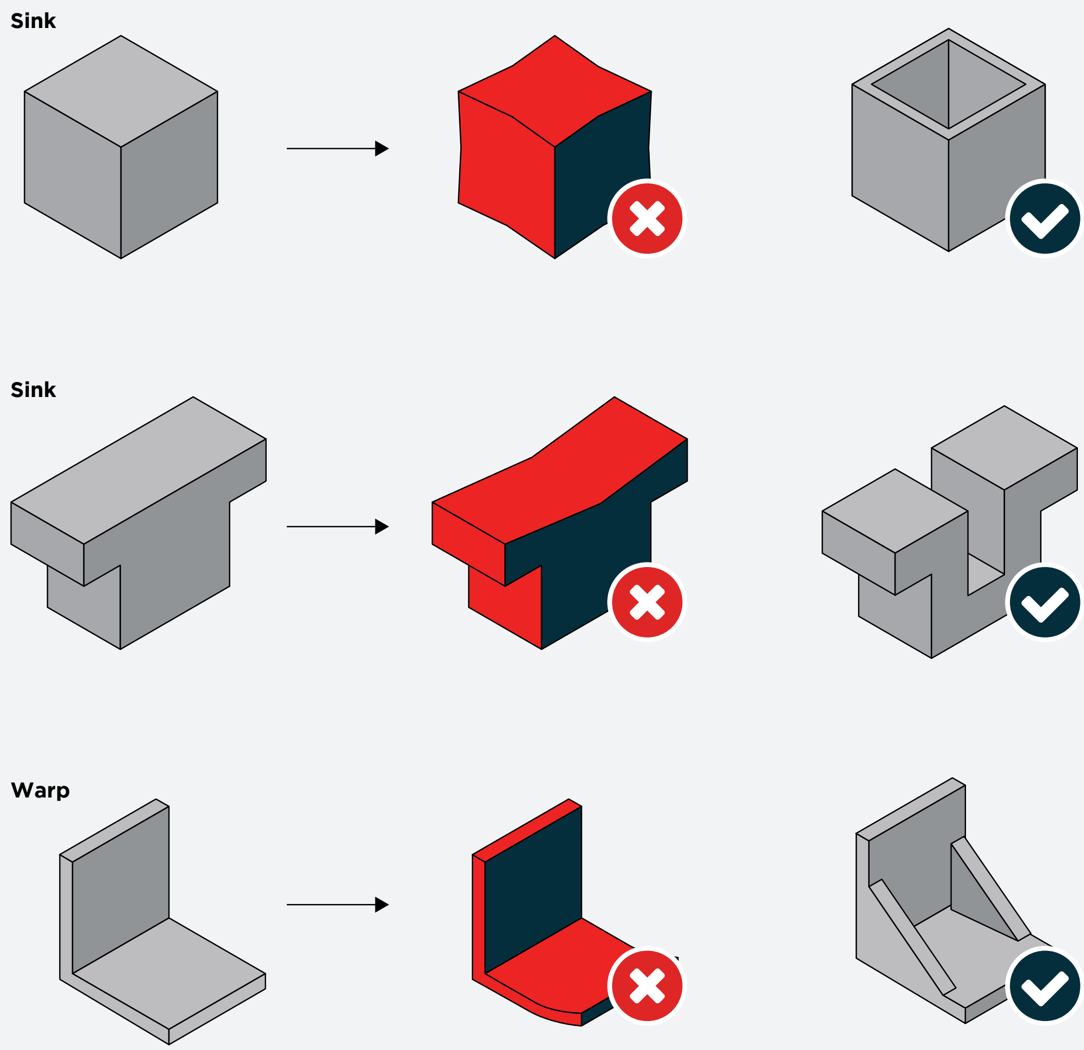
SINK & WARP

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

As the plastic material cools, the molecules move closer together. If the cooling rate differs due to wall thickness, warp may occur.

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 given in order to create high quality, consistent plastic parts.



SHRINK

A certain degree of shrinkage is expected in the injection molding process. Some materials tend to shrink more than others so careful consideration of 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.

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”

SUMMARY OF BEST PRACTICES & FEATURES

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

CORNERS AND TRANSITIONS

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

FEATURES

TEXT ON PARTS

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

HINGES & SNAPS—Used to simplify assembly, enhance function and reduce cost

THREADS

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

OVERMOLDING

- Design holes or slots in the first mold to physically interlock pieces
- 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

GETTING STARTED

STARTING YOUR PROJECT WITH XCENTRIC

> To begin a project with Xcentric, request a quote through our online portal. You will receive a response within 1 business day.

Xcentric will provide DFM review and Mold Flow Analysis during the quoting process and will provide you with feedback and suggestions on any areas that could cause problems.

Once you place your order, Xcentric will design the mold. A parting line, ejection and gate location (PEG) document will be provided for your approval. Once approved, Xcentric creates your mold and schedules your part run. With every mold ordered, Xcentric provides 25 pieces at no additional cost. We can fulfill part orders up to 250,000 pieces.

For 25 years, Xcentric has been delivering high quality custom plastic parts for a variety of industries. Utilizing proprietary processes and our advanced mold making system allows us to provide both simple and complex parts quickly.

Xcentric has gained a reputation as a trusted supplier for time-critical manufacturing services. We are able to meet our clients' milestones through:

- Streamlined quote process
- Integrated design review
- In-house manufacturing expertise
- The Xcentric process engine
- Practically no geometry limits

ABOUT XCENTRIC MOLD & ENGINEERING

MADE IN USA

Founded in 1996, Xcentric Mold & Engineering is an innovator of on demand digital manufacturing and continues to lead advances in injection molding and rapid prototyping. We know what it takes to deliver a high-quality product on time and on budget.

Ready to experience our engineer-centricity, quality, and speed for your injection molded part? Send us a project!

GET A QUOTE | TALK TO AN EXPERT: SALES@XCENTRICMOLD.COM

XCENTRICMOLD.COM | (586) 598-4636

XCENTRIC MOLD & ENGINEERING