XCENTRIC
MOLD & ENGINEERING

WHITE PAPER

5 COMMONLY OVERLOOKED CONSIDERATIONS

In Parts Design

INTRODUCTION:
Why getting parts design right is critical in the manufacturing process

Engineering leaders and product developers face many challenges in getting their products to market, and one of those is production risk. It is not just small-time manufacturers that are vulnerable. Mid to large-scale manufacturers are also exposed to the effects of regulation changes, economic uncertainty, material shortages, and consumer demand. All these factors have the power to delay or halt production, impacting a business financially. But production risk is also at ground level—challenges with internal manufacturing processes and product development phases can lead to similar losses.

Focusing on risks within your control—lead times, supplier location, optimizing the product development lifecycle, and overcoming skill shortages—is the most effective way to mitigate production risk and maximize outcomes. So, it is not surprising that 66% of CEOs are now rethinking their approach to supply chains and looking for solutions closer to home.[1] By engaging local expertise, you reduce reliance on international suppliers and product lead times. Getting support in the early phases of the product development lifecycle or where there is a lack of expertise, mitigates risks downstream.

> **66% of CEOs are now rethinking their approach to supply chains and looking for solutions closer to home[1]**

[1] KPMG's Global Manufacturing Outlook 2020

INTRODUCTION

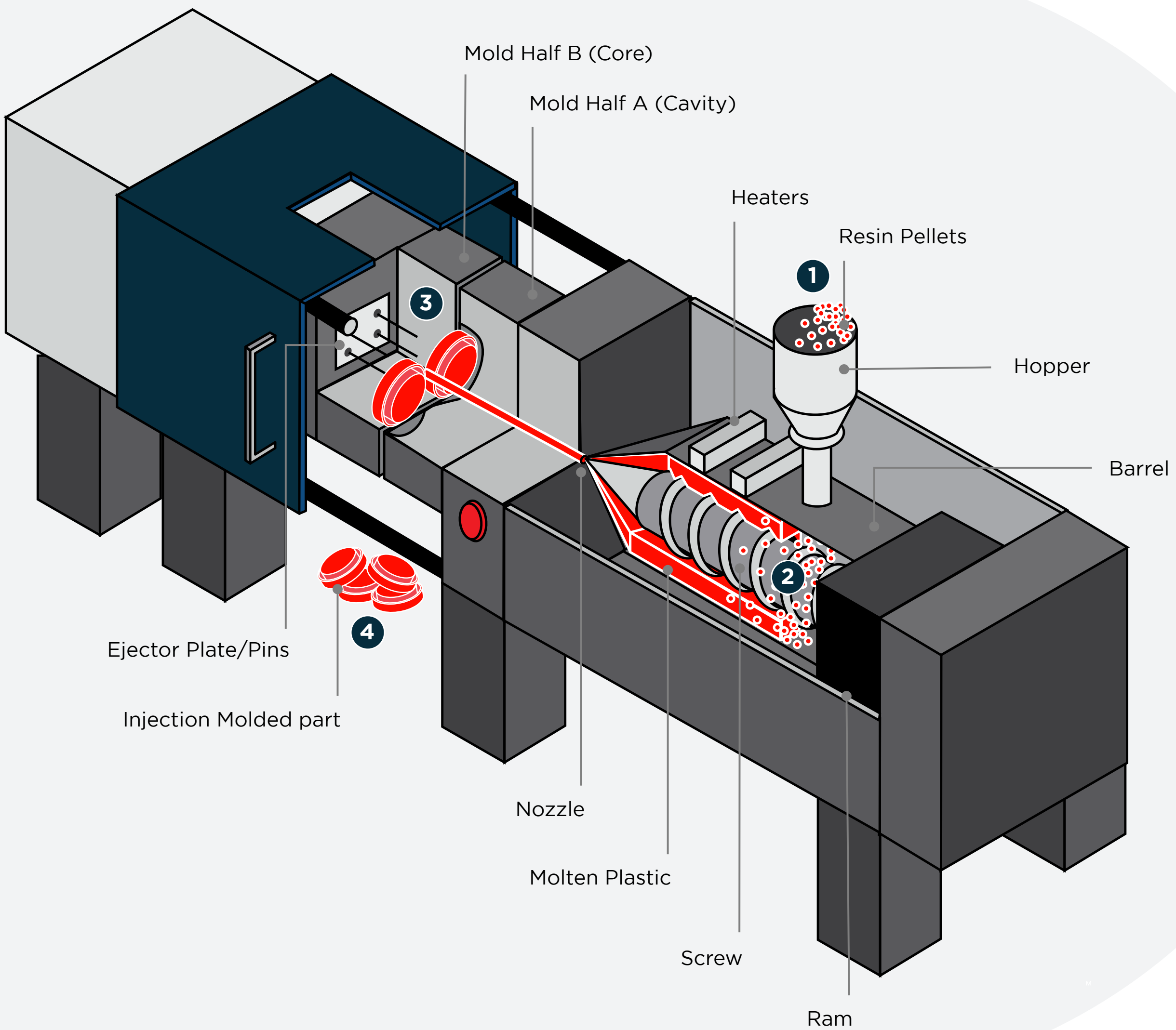
Design and prototyping are the earliest phases of the lifecycle and therefore provide you with the greatest opportunity to de-risk the pre-production and production phases. By using engineering-grade resins, prototypes can be tested under the same conditions as final parts and can be made of similar finish materials. This means your design iterations can be easily undertaken during the prototyping phase. The number of iterations that you might then require in the more cost-critical phases of the lifecycle is reduced. As a result, costs are lowered, time is recouped, and your product heads out to the market faster, ahead of competitors.

This white paper uncovers the most commonly overlooked considerations in parts design and its impact on the end-product and product development cycle. It outlines:

- **5 critical best practices to consider in your parts design process to help you minimize costs and delays**
- **The common pitfalls in parts design and how to avoid them**
- **Why having the right expertise will ensure you get your parts design right from the very start**

GETTING YOUR BASICS RIGHT

In injection molding, plastic resin pellets are loaded into the hopper of the molding machine, travel into the barrel and are then melted into molten material. When conditions inside the barrel are met, the ram moves forward driving the screw. As the screw turns, it creates pressure pushing 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.



While pressure, temperature, and time cycle are critical in creating a quality custom part, there's more to it than that. Best practice parts design takes into account the end function of your part, what it's made of, the properties required for best part performance, and finally, the cost of the part versus its benefits.

5 COMMONLY OVERLOOKED BEST PRACTICES IN PARTS DESIGN

When accuracy in parts design is compromised, it can generate a ripple effect downstream in the production process. Incorrect material selection for your part can be costly. For instance, if you neglect the mold flow analysis of your part or make a material selection solely based on price, you run the risk of issues (like knit lines) in your final part. Inadequate design clarity or skipping pivotal phases like prototyping or manufacturability analysis can lead to unforeseen costs and project delays. Understanding key considerations in design (or engaging an engineer-centric partner that does) will help you implement the right processes and optimize your design.

> **"Inadequate design clarity or skipping pivotal phases like prototyping or manufacturability analysis can lead to unforeseen costs and project delays"**

So, what's really important in designing a part for injection molding? Once you've chosen the right material/resin with your expert partner, **prioritize** these fundamentals:

1 Maintain uniform wall thickness

WHY IT'S IMPORTANT: Wall thickness determines the mechanical performance, cosmetic appearance, moldability, and cost-effectiveness of your plastic injection-molded custom parts. Getting this right minimizes expensive tooling changes down the road.

HOW TO ASSESS IT: A 10% increase in wall thickness increases the stiffness of most materials by approximately 30%. However, you can also use ribs, curves, and corrugations to help provide equivalent stiffness with less wall thickness. In this way, you can reduce material costs while still benefitting from the rigid strength and durability in your plastic molded parts.

WHAT TO AIM FOR: A balance of strength and weight.

2 Get draft correct for easy ejection

WHY IT'S IMPORTANT: Draft helps to prevent sticking and ejector pin push marks on the show surface during the molding process.

HOW TO ASSESS IT: A draft angle of 0.5˚ is the minimum draft needed for most applications. For plastic injection molding, draft angles of 1.50˚ to 20˚ per side are standard. For textured surfaces, a 3˚ to 5˚ draft angle is typically required.

WHAT TO AIM FOR: The right angle based on your mold material and texture.

3 Reinforce your structures with ribs and bosses

WHY IT'S IMPORTANT: Ribs and bosses are key allies in structure reinforcement. Ribs allow for greater strength and stiffness in molded plastic parts without the need to increase wall thickness. Bosses help with locating, mounting, and assembly.

HOW TO ASSESS IT: Ribs' thickness should be 60% of the joining wall thickness to minimize sink risks. Glossy materials, however, require a thinner rib (40% of wall thickness), but keep in mind that thin ribs can be difficult to fill. Rib height should not exceed 3x the rib-base thickness. Don't add ribs to non-critical areas. Use ribs sparingly as they can be difficult to remove.

WHAT TO AIM FOR: Optimized use of ribs and bosses—right computation, right place, right quantity.

4 Minimize use of undercuts

WHY IT'S IMPORTANT: Undercuts are any indentations or protrusions that prohibit an ejection of a part from a one-piece mold. They are used for designs that need an extra part to capture the detail as part of the mold.

HOW TO ASSESS IT: As undercuts can lead to increased mold complexity and higher mold construction costs, it is worth considering a simple re-design to identify if an undercut is truly needed. When undercuts cannot be removed, internal mold mechanisms (side-action slides, jiggler pins, lifter rails, collapsible cores, and unscrewing) may be required.

WHAT TO AIM FOR: Use undercuts only in the absence of alternatives.

5 | Maintain thickness with corners and transitions

WHY IT'S IMPORTANT: Rounded corners and tapered transitions help maintain consistent wall thickness for better part performance.

HOW TO ASSESS IT: Make the outside radius one wall thickness larger than the inside radius to keep wall thickness through the corners. If it is necessary to transition from thicker walls to thinner ones, round or taper the thickness of the transitions, so there are no abrupt stress-causing changes in design.

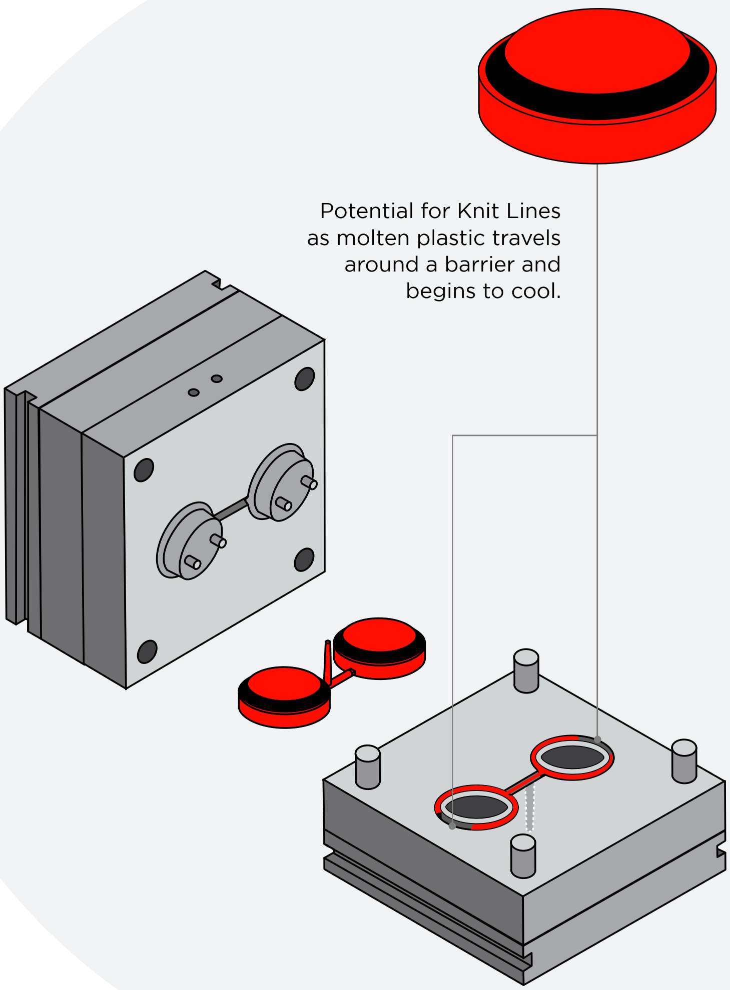
WHAT TO AIM FOR: Rounded and tapered, never sharp or abrupt.

THE PITFALLS IN PARTS DESIGN

What happens when you get parts design **wrong**?
The consequences can include:

KNIT LINES

Knit lines arise when molten plastic travels around an obstruction and its leading edge begins to cool.



HOW IT HAPPENS:

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.

MATERIAL PRONE TO KNIT LINES:

ABS

HOW TO AVOID KNIT LINES:

- Undertake mold flow analysis of your part
- Make design modifications with an expert
- Explore alternative materials less prone to knit lines

SINK AND WARP

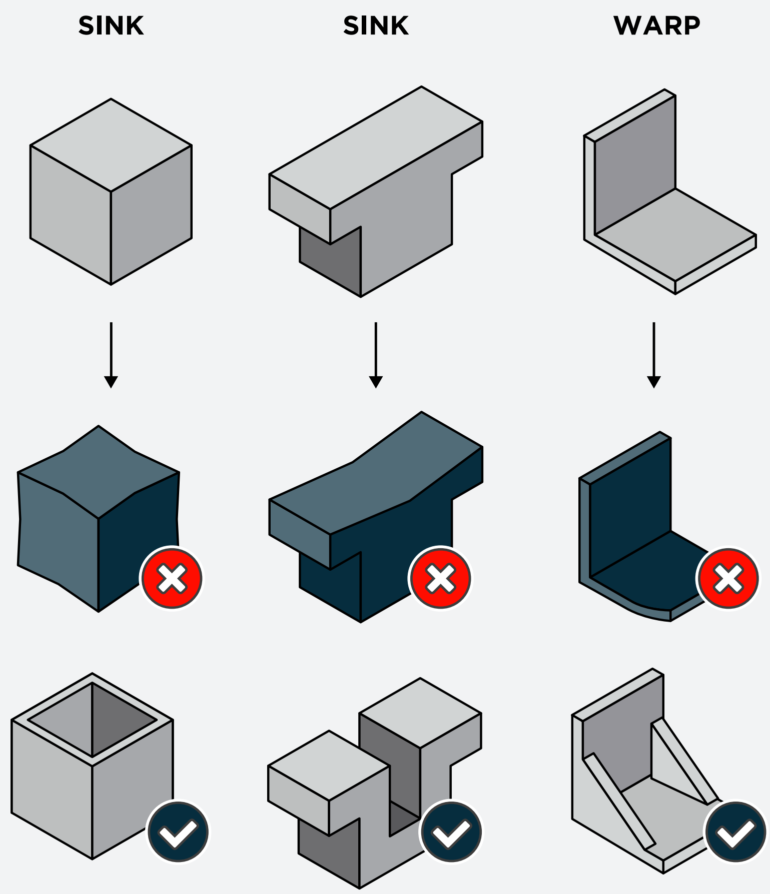
Variations of shrinkage in materials can lead to warp, distortion and dimensional issues with injection molded parts.

HOW IT HAPPENS:
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.

MATERIAL PRONE TO SINK AND WARP:
High density polyethylene (HDPE), polypropylene

HOW TO AVOID SINK AND WARP:

- Thinner and consistent wall sections



SHRINK

A certain degree of shrinkage is expected in the injection molding process.

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”

HOW IT HAPPENS:

Rapid changes to wall thickness are the most common cause.
It is caused by pressures exerted for the plastic material to fill your mold.

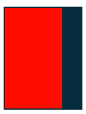
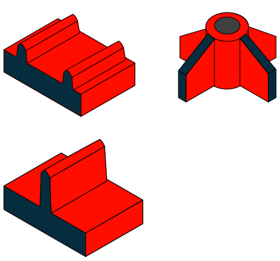
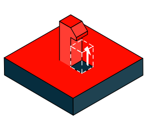
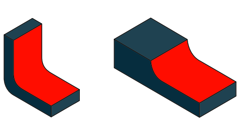
MATERIAL PRONE TO SHRINK:

Amorphous materials such as ABS, PC, and PVC have lower, more uniform shrinkage than semi-crystalline materials such as PE, nylon or polyester

HOW TO AVOID SHRINK:

- Eliminate thin wall sections leading into thicker wall sections—keep uniformity
- If necessary, transition the change gradually, utilizing angles to help aid material flow

There are risks associated with each parts design best practice, but there are ways to **manage** them.

CONSIDERATIONS	PITFALLS	RISK MITIGATION
Wall Thickness 	Sink Warp Shrink Blemishes	• Maintain uniform thickness • Utilize ribs, curves, and corrugations to reinforce walls without adding to thickness • A 10% increase in thickness = 33% increase in stiffness • Core out unneeded thickness and wall stock
Draft 	Surface damage Blemishes	• Maintain a minimum of 0.5˚ draft angle on all features perpendicular to the parting line • 1˚-2˚ is ideal
Ribs and Bosses 	Sink Shrink Warp Cosmetic issues Fill difficulty (thin ribs)	• Consider material used • Design ribs to approximately 60% of the joining wall thickness for minimum risk of sink marks • Wall thickness around bosses depend on nominal part thickness • Height of bosses should be no more than 2.3 times the diameter of boss hole
Undercuts 	Inefficient Difficult to remove	• Undercuts will add cost to the mold; minimize them when you can • Internal mold mechanisms may be required for removal
Corners and Transitions 	Molded-in stress from resin flow	• Use gradual transition if wall thickness must change • Corners: R1 + T = R2

AVOID THESE PITFALLS WITH THE RIGHT PARTNER

Getting support from a manufacturing partner with expertise in end-to-end product development cycles, complex projects, and meeting short timelines is key to parts design success. With over 25 years' experience in injection molding and prototyping, we at Xcentric know what usually gets overlooked in parts design, what the consequences are, and how to avoid them.

Our in-house molding technology is executed by our team of manufacturing experts and led by a group of seasoned US industry veterans. We provide customers with consistent results in parts design, often reducing the product to market timeframe by weeks.

"We provide customers with consistent results in parts design, often reducing the product to market timeframe by weeks."

THE XCENTRIC DIFFERENCE

MADE IN USA

1 | Engineered to be Nimble

Xcentric can deliver simple or complex parts from prototype to production, on time and on budget. Our on-demand digital manufacturing capabilities in Michigan allow us to be quick, flexible, and responsive to your design needs.

2 | Expert Team

With 25 years of industry experience, we have a robust team of experts in injection molding and rapid prototyping. Our (recently tripled) design, engineer, and customer experience teams are with you every step of the way from concept to production.

3 | Engineer Centric

We understand the day-to-day challenges engineers face, and we help alleviate those struggles. We apply an engineer-first mindset to everything we do—addressing prototype design and manufacturing complexities while ensuring projects are delivered on time and on budget.

COLLABORATION AND LOCAL EXPERTISE FOR PARTS DESIGN SUCCESS

To be successful, a collaborative process with on-hand molding experts across the full product development lifecycle is vital. We'll give you the right advice to manage your production risks, reducing the burden on your design and engineering teams. More importantly, you (and your end customers) are assured of accurate, on time, and on budget product delivery.

To find out more about how we can help you optimize your parts design, mitigate risks and minimize costs, contact us at **info@xcentricmold.com** or **(586) 598-4636**.

ABOUT XCENTRIC MOLD & ENGINEERING

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. Xcentric is engineered to be nimble, employs a team of experts in injection molding, and takes an engineer-centric approach to everything we do. Tens of thousands of product developers and engineers across North America trust Xcentric to bring their products to life.

XCENTRICMOLD.COM | (586) 598-4636

MADE IN USA

XCENTRIC MOLD & ENGINEERING