THE SMART GUIDE TO

Designing for Manufacturability

A designing for plastic injection molding resource
**Advanced Molding**

Optimizing your part design from concept through 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, and guidance to avoid common pitfalls.

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

<table>
<thead>
<tr>
<th>Injection Molding Basics</th>
</tr>
</thead>
<tbody>
<tr>
<td>04 Plastic Injection Molding</td>
</tr>
<tr>
<td>05 Mold Basics</td>
</tr>
<tr>
<td>06 Types of Molds</td>
</tr>
<tr>
<td>07 Materials / Resins</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Best Practices</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 Wall Thickness</td>
</tr>
<tr>
<td>11 Draft</td>
</tr>
<tr>
<td>12 Gates &amp; Runners</td>
</tr>
<tr>
<td>13 Critical Features</td>
</tr>
<tr>
<td>14 Ribs</td>
</tr>
<tr>
<td>15 Bosses</td>
</tr>
<tr>
<td>16 Undercuts</td>
</tr>
<tr>
<td>17 Corners &amp; Transitions</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Features to Incorporate</th>
</tr>
</thead>
<tbody>
<tr>
<td>19 Text on Parts</td>
</tr>
<tr>
<td>20 Hinges &amp; Snap Features</td>
</tr>
<tr>
<td>21 Threads</td>
</tr>
<tr>
<td>22 Overmolding</td>
</tr>
<tr>
<td>23 Insert Molding</td>
</tr>
<tr>
<td>24 Surface Finishes</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Avoiding Pitfalls</th>
</tr>
</thead>
<tbody>
<tr>
<td>26 Knit Lines</td>
</tr>
<tr>
<td>27 Sink &amp; Warp</td>
</tr>
<tr>
<td>28 Shrink</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Summary</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 Best Practices &amp; Features</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Getting Started</th>
</tr>
</thead>
<tbody>
<tr>
<td>32 Starting Your Project with Xcentric</td>
</tr>
</tbody>
</table>
Injection Molding Basics

The Basics of Molds and the Plastic Injection Molding Process
Plastic Injection Molding

To understand part design, learning the injection molding process is essential. 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.
Types of Molds

PROTOTYPE MOLDS
Prototype molds are usually built from aluminum, enabling shorter build times than production molds and facilitating quick modifications should the injection molding process or the part require them.

Producing prototype parts quickly will help you to get your products to market faster than your competition. 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 approach enables you to test in real mechanical, chemical and environmental circumstances and help you create the best possible part design for your product.

BRIDGE MOLDS
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.

PRODUCTION MOLDS
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. Lessons learned through the prototyping process are incorporated into the design of the production tools.
Materials / Resins

Material selection will be one of the first and most important steps of designing your part. Before you begin, consider your part’s end function. Ensure the properties required for performance and cost of material are optimal.

Producing high quality, consistent plastic injection molded parts relies heavily on your chosen material. Visit us at: www.xcentricmold.com/plastics/ 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 technical specialists.

There are currently 62,000+ thermoplastic resins to choose from and these resins are available in a wide assortment of grades with different properties. For that reason, we recommend you visit www.matweb.com 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.

- Off-the-shelf colors are generally less expensive than custom colors
- Your injection molding partner should be able to source the material you need

EXAMPLES OF ADDITIVES TO CONSIDER:

- **GLASS FIBER** — 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

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
- Polymers
- Polycarbonate
- Polyurethane
- Polyphenylene (PPE)
- Polyvinylidene (PVDF)

STANDARD RESINS

185°F
- Polypropylene
- Polyethylene
- ABS Plastics
## Materials / Resins

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

<table>
<thead>
<tr>
<th>Material</th>
<th>Mechanical Properties</th>
<th>Moldability Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Strength</td>
<td>Hi Temp Strength</td>
</tr>
<tr>
<td>Acrylic</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>ABS Plastic</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>Acetal</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>Thermo-Elastomer</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>High Density Polyethylene (HDPE)</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>Nylon 6/6</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>Nylon 6/6 (Glass-filled)</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>Polybutylene (PB)</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>Polycarbonate (PC)</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>Polybutylene and Polyethylene</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>Polypropylene</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
<tr>
<td>Polystyrene</td>
<td><img src="icon.png" alt="Low" /></td>
<td><img src="icon.png" alt="Low" /></td>
</tr>
</tbody>
</table>
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, 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 to 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 27
Shrink — page 28

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>RECOMMENDED WALL THICKNESS (INCHES)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABS Plastic</td>
<td>0.045 – 0.140</td>
</tr>
<tr>
<td>Acetal</td>
<td>0.030 – 0.120</td>
</tr>
<tr>
<td>Acrylic</td>
<td>0.025 – 0.500</td>
</tr>
<tr>
<td>Liquid Crystal Polymer</td>
<td>0.030 – 0.120</td>
</tr>
<tr>
<td>Long-fiber Reinforced Plastic</td>
<td>0.075 – 1.00</td>
</tr>
<tr>
<td>Nylon</td>
<td>0.030 – 0.115</td>
</tr>
<tr>
<td>Polycarbonate (PC)</td>
<td>0.040 – 0.150</td>
</tr>
<tr>
<td>Polyester</td>
<td>0.025 – 0.125</td>
</tr>
<tr>
<td>Polyethylene</td>
<td>0.030 – 0.200</td>
</tr>
<tr>
<td>Polyethylene Sulfide</td>
<td>0.020 – 0.180</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>0.025 – 0.150</td>
</tr>
<tr>
<td>Polystyrene</td>
<td>0.035 – 0.150</td>
</tr>
<tr>
<td>Polyurethane</td>
<td>0.080 – 0.750</td>
</tr>
</tbody>
</table>
**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.
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 28).
• 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 structure 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 27) 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**

<table>
<thead>
<tr>
<th>RESIN</th>
<th>MINIMAL SINK</th>
<th>SLIGHT SINK</th>
</tr>
</thead>
<tbody>
<tr>
<td>PC</td>
<td>50% (40% if high gloss)</td>
<td>66%</td>
</tr>
<tr>
<td>ABS</td>
<td>40%</td>
<td>60%</td>
</tr>
<tr>
<td>PC/ABS</td>
<td>50%</td>
<td>66%</td>
</tr>
<tr>
<td>Polyamide (Unfilled)</td>
<td>30%</td>
<td>40%</td>
</tr>
<tr>
<td>Polyamide (Glass-Filled)</td>
<td>33%</td>
<td>50%</td>
</tr>
<tr>
<td>PBT Polyester (Unfilled)</td>
<td>30%</td>
<td>40%</td>
</tr>
<tr>
<td>PBT Polyester (Filled)</td>
<td>33%</td>
<td>50%</td>
</tr>
</tbody>
</table>
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.
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 logo’s, 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 point 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.
- 20 point or larger text.
- 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 inch in width and at least .008 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 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 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.
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 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 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 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 cosmetics to your finished plastic injection molded parts. Finishes can give a mirror-like gloss finish or a textured finish for grip and usability.

COMMON SURFACE FINISHES:
- 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)

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 primer.
Avoiding Pitfalls

Unexpected Issues Can Occur with 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 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.

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>SHRINK INCH / INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polypropylene - Unfilled</td>
<td>0.015” - 0.018”</td>
</tr>
<tr>
<td>Polyethylene</td>
<td>0.020” - 0.025”</td>
</tr>
<tr>
<td>ABS</td>
<td>0.0035”</td>
</tr>
<tr>
<td>HIPS</td>
<td>0.0035”</td>
</tr>
<tr>
<td>GPPS</td>
<td>0.0035”</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>0.007”</td>
</tr>
<tr>
<td>PC-ABS</td>
<td>0.007”</td>
</tr>
<tr>
<td>Acrylic</td>
<td>0.003” - 0.004”</td>
</tr>
<tr>
<td>Nylon 6/6 (PA66) - Unfilled</td>
<td>0.020”</td>
</tr>
<tr>
<td>Nylon 6/6, 33% Short Glass</td>
<td>0.0035”</td>
</tr>
<tr>
<td>PBT</td>
<td>0.015”</td>
</tr>
<tr>
<td>Acetal (POM)</td>
<td>0.020”</td>
</tr>
<tr>
<td>Acetal Homopolymer</td>
<td>0.018” - 0.020”</td>
</tr>
<tr>
<td>PVC (Rigid)</td>
<td>0.0035”</td>
</tr>
<tr>
<td>TPE (Santoprene)</td>
<td>0.014” - 0.018”</td>
</tr>
<tr>
<td>Noryl</td>
<td>0.005” - 0.007”</td>
</tr>
<tr>
<td>Noryl 30% Glass Filled</td>
<td>0.001”</td>
</tr>
<tr>
<td>TPU</td>
<td>0.007” - 0.010”</td>
</tr>
<tr>
<td>Polysulphone</td>
<td>0.007”</td>
</tr>
</tbody>
</table>
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 100,000+ pieces.

For over 20 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’ time critical milestones through:

- Streamlined Quote Process
- Integrated Design Review
- In-house Manufacturing Expertise
- The Xcentric Process Engine
- Practically No Geometry Limits