Designing for Plastics: Optimizing Your Injection Molded Parts
In the mid 90's, two Michigan-bred brothers working for other companies as mold makers realized that all the mold shops in the Detroit area lacked technology and relied primarily on traditional and more manual processes. In 1995, they decided that the only way to differentiate from their peers in the trade was to learn and apply technology on their own and Xcentric Mold & Engineering was born.

They built a proprietary advanced technology platform focused on software, automation, systems, and custom standardization and processes – it would become the foundation that would surpass all other traditional manufacturing companies. Employees were hand-picked and trained through a rigorous advanced mold making program that the founders created so each individual could speak technically and with confidence to customers on the fly. The art of mold making was transformed into the science of custom quick-turn manufacturing and Xcentric Mold & Engineering of today where quality, design and speed are not compromised.

The brothers knew they created something special. Because of their vision and strong belief in utilizing and creating technology, processes and systems along with building a team of professionals, Xcentric was able to compete even when jobs were being lost to lower labor countries overseas. And when the bottom fell out in the economy in 2009 it had very little effect on the company. A story of two brothers, a vision realized through courage and innovation combining the art and science of mold making, leading to over 20 years of experience as Xcentric Mold & Engineering.
Xcentric Mold & Engineering provides the fastest, most affordable quick-turn low-volume injection molding, CNC machining and rapid prototyping services within 15 days or less. Xcentric can manufacture complex parts without compromising the engineers’ design or time-to-market.

**Features:**
- Simple and Complex Insert Molding
- Simple and Complex Overmolding
- Unlimited Undercuts
- Internal & External Threading
- Free Manufacturability Review

Xcentric complement’s their automated manufacturing through a skilled sales engineering team who works with design engineers to optimize their parts. In addition, we achieve tighter tolerances through our quality control tooling process.

**Highlights:**
- 20 years in business
- Two Highly Advanced Manufacturing Facilities Located in South East Michigan
- All molds are built and run onsite at each facility
- Over 10,000 tooling orders placed
- Life Time Mold Guarantee
- 1,000’s of different engineered grade materials processed
- 50+ industries served
- Proprietary advanced manufacturing processes and automation
- Manufactured in the USA
Before we jump into designing for plastics, here’s a quick review of the Injection Molding Process and Rapid Injection Molding.

**Injection Molding 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.

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 channeling the molten plastic into the mold cavity through the nozzle. Once allowed to cool, the mold will open and the ejection plate engages, releasing the final part from the mold.

**Rapid Injection Molding**

Rapid injection molding is possible when a manufacturer has developed sophisticated digital manufacturing practices 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, you can 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. Done correctly, you can often skip rapid prototyping because the process is so fast and affordable allowing you to produce low-volume production quality parts.
No matter which manufacturing process you select for your project, the most important question you can ask yourself is “How will the part be used?".

**Concept or Final Production** – one end of the continuum demands that you are in pilot mode manufacturing multiple prototypes to determine the best design, while the other end of the continuum demands efficient manufacturability alignment with the intent to flawlessly go-to-market.

**Part Simplicity or Complexity** – for simple parts build a list of non-negotiables that the manufacturer must adhere to, as not all parts can be manufactured as you intended even though the design is “simpler”. For complex parts involve your manufacturer early to improve your design’s manufacturability and potential approaches that could remove complexity and cost out of your design.

The success of your plastic part relies upon many key elements working simultaneously. There are many factors that influence the outcome of your plastic part and should be taken into consideration when designing with plastics. The following pages provide a reference to each key element:

• Draft Angles
• Wall Thickness
• Fillets and Radii
• Bosses
• Ribs
• Holes/Depressions
• Shrinkage
• Gating
• Vents
• Potential Knit Lines
Draft Angles

- A typical draft angle is 1° with 1/2° on ribs
- Some draft is better than none
- With minimum draft, fine polish mold in direction of ejection
- Allow 1° additional draft for each .001 inch depth in texture

Wall Thickness

- Wall thickness should be kept uniform
  - Provides even melt flow and uniform shrinkage
- Core out thick sections
  - Provides uniform shrinkage and faster cycle time
- Changes in wall thickness should be transitioned
  - Avoids stress concentrations and cooling differences
- Gate into thick sections
  - Provides improved packing and reduces stresses

The number one rule for designing with plastics is uniform wall thickness.
Fillets and Radii

- Fillet radius should be 25-60% of nominal wall thickness
- Avoids stress concentrations
- Outside corner radius should equal inside radii plus the wall thickness
- Helps keep uniform wall thickness and avoid stress concentrations
- Inside radius = 0.5 x wall thickness
- Outside radius = 1.5 x wall thickness
- Sharp corners are the #1 cause of part failure, stress concentrations, poor flow patterns and increased tool wear. Sharp corners should always be avoided!

Bosses

- Bosses are projections designed to accept inserts, self tapping screws, etc. for use in mounting, fastening or assembling parts
- Typically the boss O.D. = 2 I.D.
  - Allows the wall to increase as the size of the boss increases
- Wall thickness at base of boss < 60% of nominal wall
  - Reduces sink marks and cycle time
- Boss height < 3 x O.D.
  - Minimizes excess mass at bottom of boss and allows for sufficient cooling of core pin
- Base of boss should be radiused 25 –50% on nominal wall (minimum of .015-inch)
  - Minimizes stress concentration in high stress area
Ribs, Holes and Depressions

Ribs

- Base of rib should be 50 –60% of nominal wall thickness
  - Avoids sink marks and excess mass
- Maximum rib height of 3 times the nominal wall thickness
  - Deep ribs are hard to fill, may stick in mold
- Draft of 1° -1.5° is desirable
  - Minimum draft of 1/2° is required for ejection
- Base of rib should be radiused 25 –50% on nominal wall thickness
  - Minimizes stress concentration in high stress area
- Spacing between 2 parallel ribs should be minimum of 2 times nominal wall thickness
  - Avoids hot spots on mold
- The preferred flow of melt is down the length of the ribs
  - Flow across ribs can result in trapped gas

Holes and Depressions

- Blind holes: L/D ratio < 2
  - Helps minimize deflection of core pins
- Through holes: L/D ratio < 4
  - Because core pins can be supported at both ends the L/D ratio can be longer
- The melt should flow down the length of a slot or depression
  - Helps minimize stresses and improves knit line strength
- Distance between two holes or one hole and the edge of the surface should be 2 times the wall thickness or 2 times the hole diameter, whichever is larger
  - Provides adequate material for strength, especially if knit lines are present
- Edges and corners of depressions should have a minimum radius of .015-inch
  - Aids in molding and strength of the part
- Base of rib should be 50 –60% of nominal wall thickness
  - Avoids sink marks and excess mass
- Maximum rib height of 3 times the nominal wall thickness
- Draft of 1° -1.5° is desirable
  - Minimum draft of 1/2° is required for ejection
- Base of rib should be radiused 25 –50% on nominal wall thickness
  - Minimizes stress concentration in high stress area
- Spacing between 2 parallel ribs should be minimum of 2 times nominal wall thickness
  - Avoids hot spots on mold
- The preferred flow of melt is down the length of the ribs
  - Flow across ribs can result in trapped gas
Shrinkage, Gating, and Vents

Shrinkage

- Shrinkage is an inherent property of the material
- Amorphous materials have lower, more uniform shrinkage than semi-crystalline materials
- Processing conditions, especially mold temperature and pack pressure, can affect shrinkage
- Gate size and design, as well as part and mold design will affect shrinkage

Gating

- Gate thickness should be 50 –80% of the wall thickness. Gate thickness controls packing time.
- Gate width should be twice the gate thickness
- Gate lands should be kept to a minimum
- Gates should be located at right angles to the runner to allow the polymer melt to impinge against the mold wall
- Gates should be located in thick sections of the part

Vents

- Allow air to escape from the mold cavity as the plastic melt flows in
- Help prevent burning or dieseling
- Help prevent mold deposits
- Help prevent incomplete mold fills
- Help prevent slowing down the injection speed
- Amorphous materials can tolerate deeper vents than semi-crystalline materials
- Vents should be ‘draw-polished’ in direction of flow to be self-cleaning
- Runners should be vented at any change of direction
- Typical Dimensions:
  \[ L = .03 - .06 \text{ inch} \]
  \[ W = .375 - .5 \text{ inch} \]
  \[ D1 = .0005 - .003 \text{ inch} \]
  \[ D2 = .01 - .04 \text{ inch} \]
Part Design Summary

Bosses
- Boss height should be 2.5 times the I.D. of the boss.
- Separate the bosses from the walls in order to maintain a uniform wall.
- The receiving hole diameter of a boss should be equal to the self-tapping screw pitch diameter.
- The boss O.D. should be equal to twice the insert or screw major diameter.

Ribs
- Reinforcing ribs should not be thicker than 2/3’s the nominal wall thickness. Any thicker will create sink marks.
- The distance between two ribs should not be less than two times the nominal wall.
- A strengthening gusset is typically two times the wall thickness deep, four times the wall thickness tall, and half the wall thickness.

Holes
- Expect a flow line on a smooth surface when the plastic flow came from a textured surface.
- All holes will create a weldline.
- Any depression that passes 2/3’s of the way through the wall creates a weldline.
- Irregularly shaped or square holes are not recommended.
- The space between two holes should be the greater of two times the wall thickness or two times the hole diameter.

Walls
- Walls should have a uniform thickness throughout the whole design; core out thick sections.
- Must have draft — a good starting point is at least 1 degree of draft per inch. Any draft angle is better than no draft angle.
- Texture — add 1 degree of draft for .001 inch of texture depth.
- If nominal wall thickness must change, the change should be gradual and should not exceed 15% of the nominal wall.

Radius
- No sharp corners: radius all inside and outside corners.
- The radius of the inside corner of a part should be 50% of the nominal wall.
- The radius of an outside corner of a part should be 150% of the nominal wall.
- Radii can be as large as the design can allow.
- Radii are impractical in certain mold locations such as parting lines or side-cored shutoffs.
- Radius a minimum of 25% of nominal wall at the junction of walls and reinforcing ribs.

Weld Lines
- Parts with weld lines are weaker than parts without weld lines.
Additional Resources

Xcentric can provide you additional resources to increase your level of understanding of injection molding and manufacturability. Click the images to view in your browser.

Webinar Series

White Paper: The Smart Guide to Designing for Manufacturability

Video Tour

About Injection Molding

Consider us for a project you are working on today. We will provide a quote within 24 hours or less.

Get a Quote