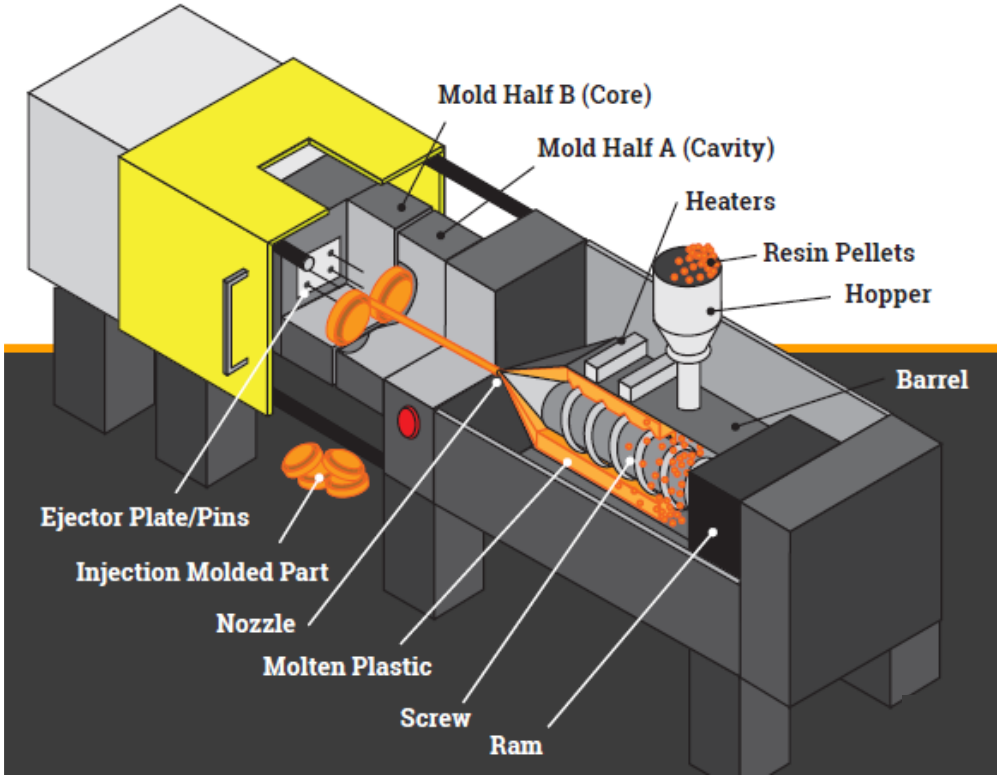
# Designing for Manufacturability
2019

XCENTRIC MOLD & ENGINEERING 24541 Maplehurst Drive | Clinton Township, MI 48036 | 586-598-4636 | www.xcentricmold.com

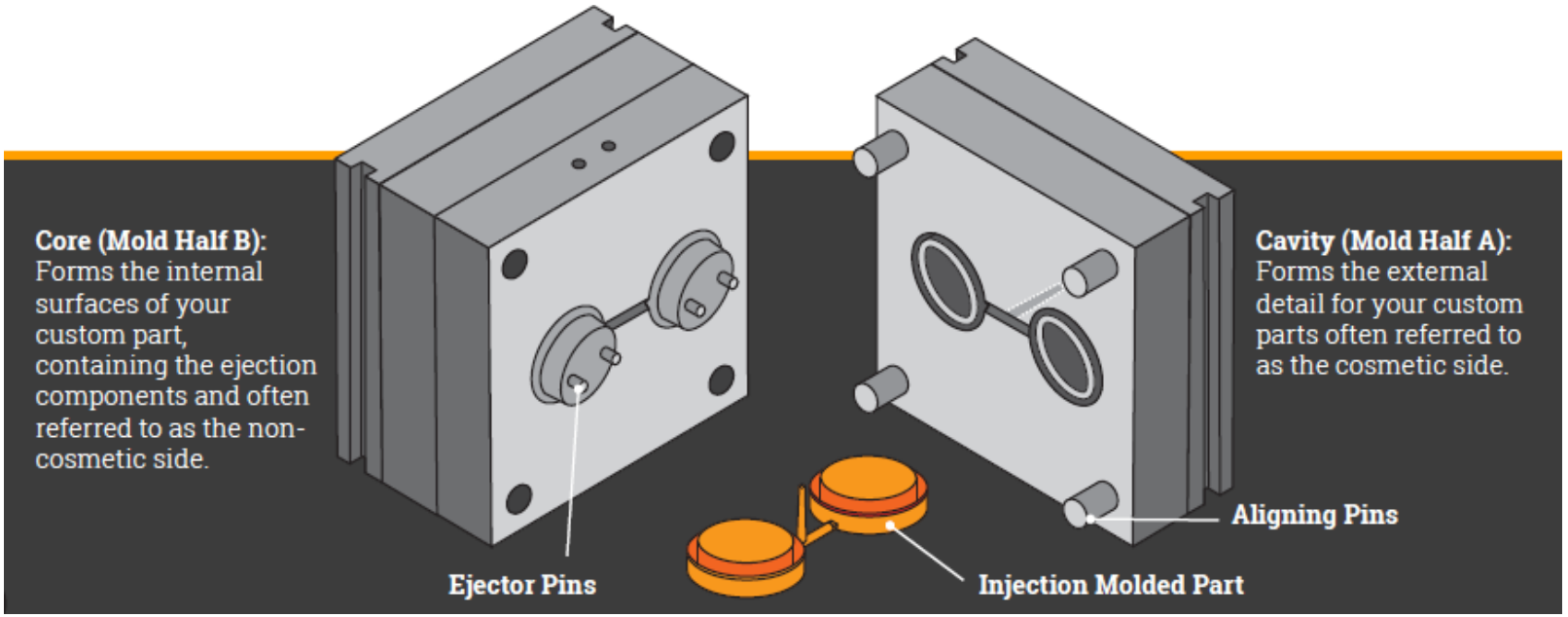
# Injection Molding Selection Criteria

- Why choose injection molding?
  - ✓ Design integrity
  - ✓ Part tolerances
  - ✓ Form-fit-function
  - ✓ Iteration cycle
  - ✓ Cost
  - ✓ Repeatability
  - ✓ Scale
  - ✓ Prototype-to-production
  - ✓ Time-to-market



# Injection Molding Fundamentals

- High quality, efficient tooling relies heavily on good part design as well as advanced skills in mold design and the manufacturing of the tool.



# Materials

- You must consider your parts end function and what properties are required to ensure that the performance and cost of material is optimal.

| | Mechanical Properties | | | Moldability Properties | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| | Low | Average | High | Poor | Average | Good | Great | | |
| 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 | $$ |
| Polybutylene (PB) | Average | Low | High | Good | Good | Average | Good | Good | $$$ |
| Polycarbonate(PC) | Average | High | High | Good | Good | 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 | $ |

# Tight Tolerances

- Unfortunately, that term is thrown about loosely. If not performed correctly, a tight tolerance part can lead to loss in performance or even part failure.
  - ✓ Utilize low-shrinkage materials for parts with tight tolerances
  - ✓ Avoiding tight tolerance areas around the alignment of the mold halves (parting line) or moving mold components such as slides
  - ✓ Design your parts to avoid tight tolerance in areas prone to warpage or distortion

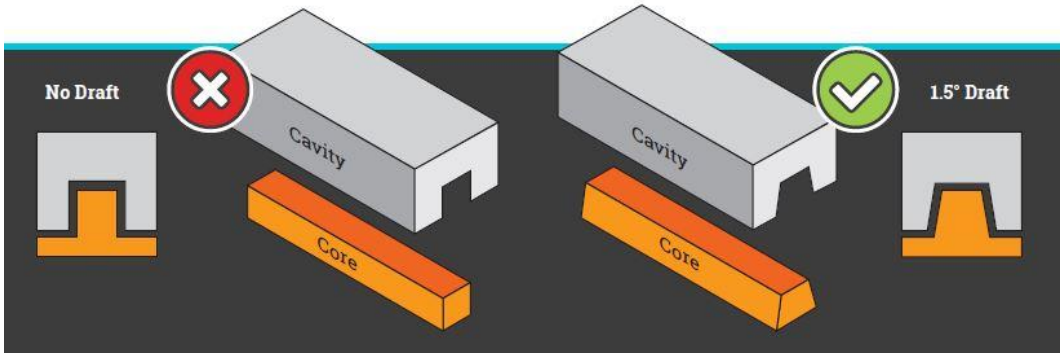
# Wall Thickness

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

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

# Draft

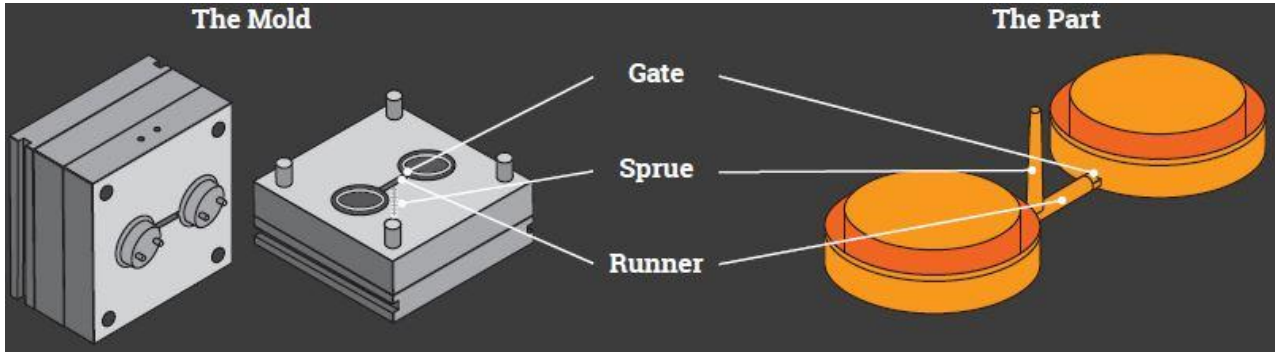
- Parts should be designed with draft to prevent sticking and ejector pin push marks on the show surface during the molding process.




- ✓ A draft angle of 0.5° is the minimum draft needed for most applications
- ✓ Draft angles of 1.5° to 2° per side are standard for plastic injection molding
- ✓ For surfaces that will be textured, a 3° - 5° draft angle is required

# Runners 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.



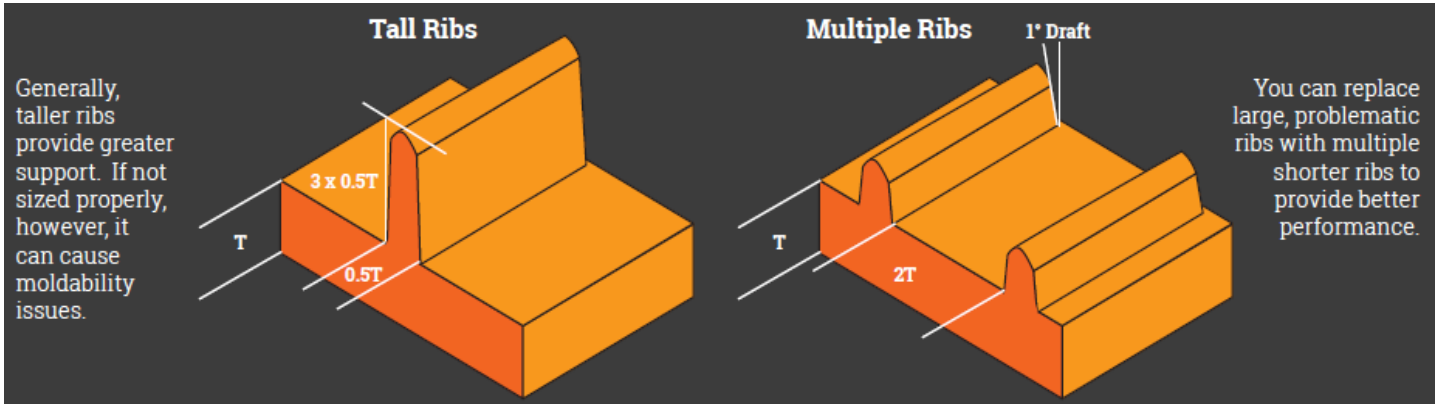

*Runners and gates control the flow of the molten material through the mold and into the cavity to create your final plastic part*

# Ribs

- Ribs allow greater strength and stiffness in molded plastic parts without the need to increase the wall thickness.
  - ✓ Design ribs that are approximately 60% of the joining wall thickness for minimum risk for sink marks
  - ✓ Glossy materials require a thinner rib (40% of wall thickness)

**Rib Thickness as a percentage of wall thickness**

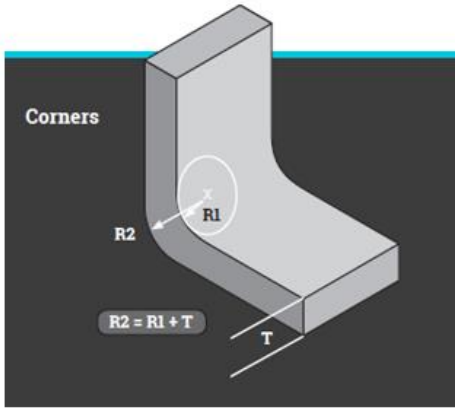
| Resin | Minimal Sink | Slight Sink |
|---|---|---|
| PC | 50%(40% if high gloss) | 66% |
| ABS | 40% | 60% |
| PC/ABS | 50% | 66% |
| Polyamide (Unfilled) | 30% | 40% |
| Polyamide (Glass-Filled) | 33% | 50% |
| PBT Polyester (Unfilled) | 30% | 40% |
| PBT Polyester (Filled) | 33% | 50% |



# Bosses

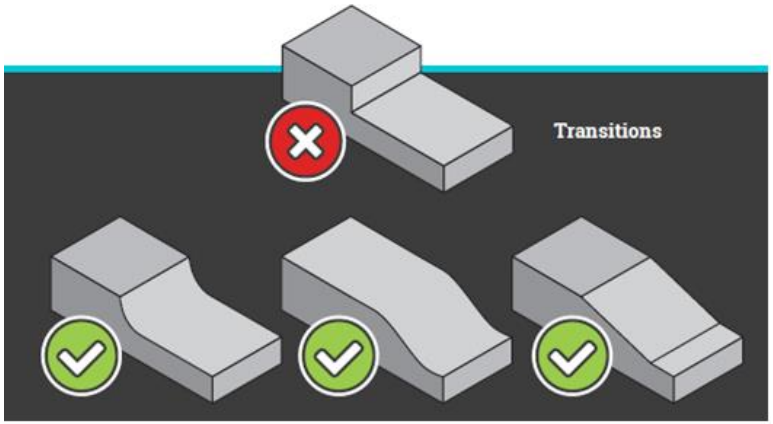
- Bosses are used for locating, mounting and assembly - wall thickness and height are the biggest factors.
  - ✓ Wall thickness around a boss design feature should be 60% of the nominal part thickness, if that thickness is less than 1/8”
  - ✓ The height of the boss should be no more than 2-1/2 times the diameter of the hole in the boss

# Corners



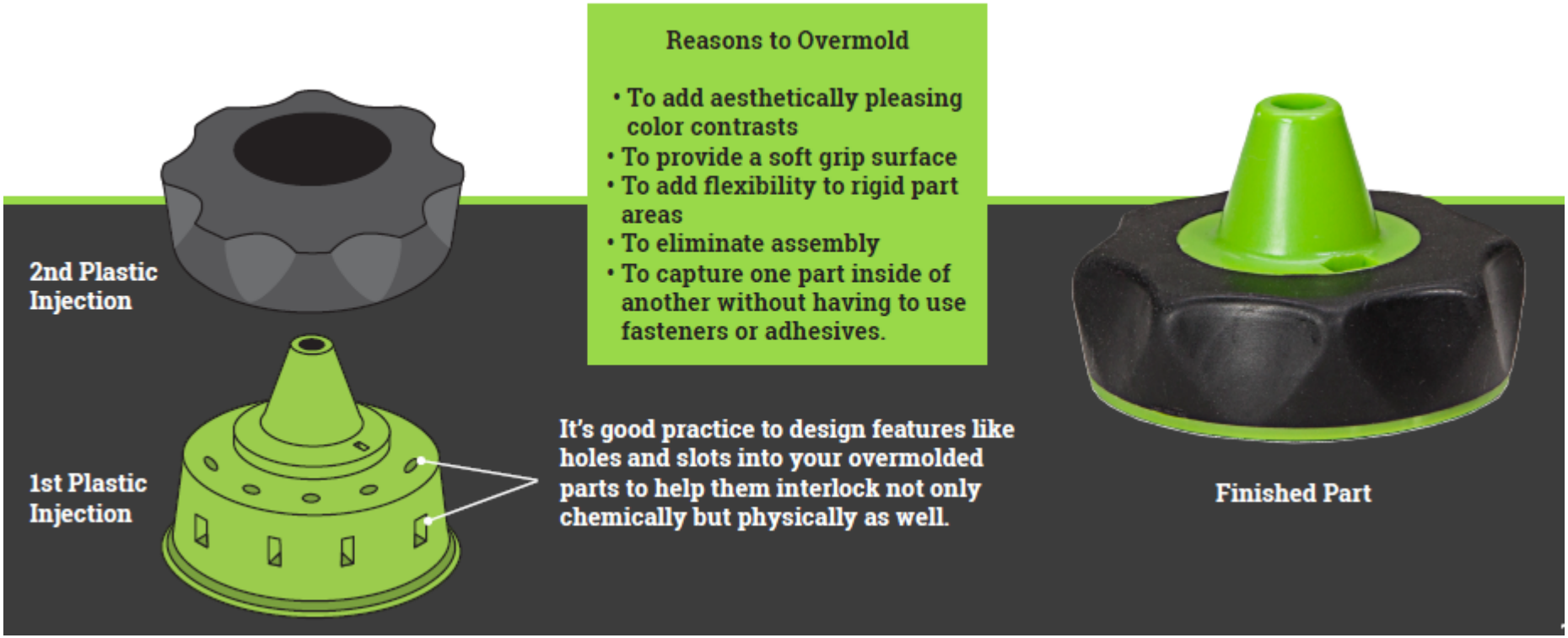

- ✓ Sharp corners can cause molded-in stress from resin flow. It is important to minimize this stress by using rounded corners which also helps to maintain consistent wall thickness

- ✓ Round or taper the thickness of your transitions to minimize molded stresses and stress concentration



# Overmolding

- Overmolding plastic parts can help in wide range of functional and structural uses. A wide range of materials are capable of being overmolded, including both hard and soft plastic resins.



# Insert Molding

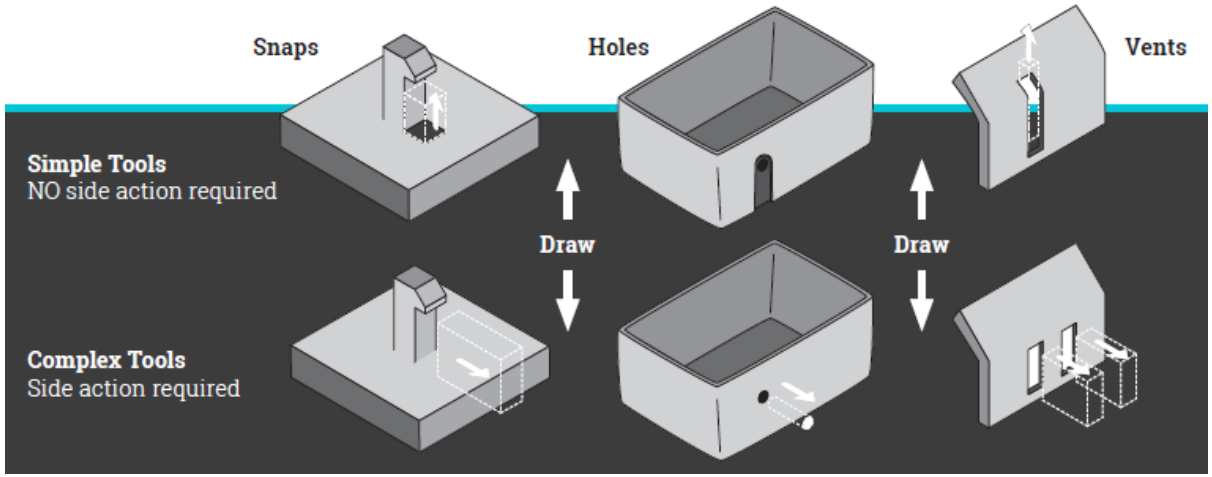
- Insert molding is the process of injection molding molten thermoplastic around pieces placed in the injection molding cavity resulting in a strong bond between integral pieces of your final part.

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

# Undercuts

- An undercut is any indentation or protrusion that prohibits an ejection of a part from a one-piece mold, usually leading to higher costs.

- ✓ When possible, design your part to eliminate undercuts as they will add cost to the mold
- ✓ Mold makers may not be advanced enough and will put limits on undercuts



# Best Practice Checklist

- ✓ Adhere to best practices in designing your part for the plastic injection molding process. Consider the following "checklist" as a baseline to meeting your part expectations.

## BEST PRACTICES

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

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

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

**Tight Tolerances**
- Utilize low-shrinkage materials for parts with tight tolerances

**Ribs & Bosses**
- Design ribs and bosses to approximately 60% of the joining wall thickness for minimum risk for sink marks.

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

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

# Q&A

# Contact Us

**Call:** (586) 598-4636

**Email:** sales@xcentricmold.com

**Web:** www.xcentricmold.com

# Visit Us

## **STATE-OF-THE-ART FACILITIES** WITH ADVANCED ROBOTICS


### HEADQUARTERS & MANUFACTURING PLANT #1

**24541 Maplehurst**
**Clifton Twp., MI 48036**
**(586) 598-4636**

### SECOND FACILITY MANUFACTURING PLANT #2

**50610 Sabrina Drive**
**Shelby Twp., MI 48315**

# Thank You!