9 Factors to Consider in Plastic Molded Design

K&B
Molded Products

Table Of Contents

Introduction

K&B Molded Products is an award-winning full service manufacturer of thermoplastic injection molded parts. We provide superior tooling and value added services as well. We drew upon our 50 years of experience to create a guide that will help you create a product with the same uncompromising quality and precision that we provide our customers.

Design Considerations

1. DRAFT

What good is a well-designed part if you can't get it out of the mold without distorting or damaging it? Draft features facilitate part removal. Though important in any production volume, they are critical with high speed operations. This makes implementing draft elements an essential design consideration. Injection molds routinely operate under high pressure and this means that the plastic comes into intimate contact with all of the mold's surfaces. Without well-designed draft, ejecting the part is exceptionally difficult.

Draft angles are not a one-size-fits-all calculation. They must be customized for each part, as some part configurations require more draft than others. Likewise, material matters. If the plastic being used has a lower coefficient of friction, less draft may be used. Whereas, an abrasive material that is more rigid requires a greater amount of draft. Of course you must also bear in mind the appearance and function of the part when draft angles are incorporated. (If no draft is acceptable due to design considerations, a side action mold may be the only option.) Another consideration when determining the amount of draft required is whether or not your part will be textured as adding texture to a part will generally increase the amount of draft required so as not to scuff the part as it ejects from the mold.

Design Considerations

2. SURFACE FINISH

When you design a part for injection molding, it is important to consider how the surface finish will affect moldability, cost, and lead time. Some finishes cost more and take additional days to be applied to your product.

The industry standard finishes created by the Society of the Plastics Industry include:

GLOSSY SURFACE, DIAMOND BUFF POLISH	
SPI Finish A-1	Grade #3, 6000 Grit Diamond Buff
SPI Finish A-2	Grade #6, 3000 Grit Diamond Buff
SPI Finish A-3	Grade #15, 1200 Grit Diamond Buff
NON-GLOSSY SURFACE, PAPER POLISH	
SPI Finish B-1	600 Grit Paper
SPI Finish B-2	400 Grit Paper
SPI Finish B-3	320 Grit Paper
ROUGH SURFACE, STONE POLISH	
SPI Finish C-1	600 Grit Stone
SPI Finish C-2	400 Grit Stone
SPI Finish C-3	320 Grit Stone
VERY ROUGH SURFACE, DRY BLAST POLISH	
SPI Finish D-1	600 Stone Prior to Dry Blast Glass Bead #11
SPI Finish D-2	400 Stone Prior to Dry Blast #240 Oxide
SPI Finish D-3	320 Stone Prior to Dry Blast #24 Oxide

Design Considerations

3. WITNESS LINES

Witness lines are an unavoidable result of uniting two mold components. They can happen along a parting line or where a core pin seals off against a mold feature like a slide. It is important that your design avoids witness lines in critical areas because they not only have an effect on the part aesthetically, but they also often affect the part's performance.

Flash, often also referred to as "burr," is excess material that goes beyond the intended geometry of the part. It is more likely to appear where witness lines are present and flash can interfere with performance by interfering with its ability to pair with other parts. This is another reason to design carefully and keep witness lines outside critical areas.

4. WALL THICKNESS

Wall thickness is yet another important factor in part design for various reasons. For one, cost savings will be highest when your part has minimum wall thickness since you will be using less plastic. (Of course the walls must be thick enough for the part to serve its function.) Likewise, parts with thinner walls cool faster and so cycle times are shorter, allowing for more products to be created in a shorter amount of time. If you need thicker walls for structural reasons, be prepared for longer cycle times and greater shrinkage during cooling than you would see in parts with thin walls.

Design Considerations

If your part will allow it, uniformity in wall thickness is ideal. This allows for greater ease in filling the mold cavity since the molten plastic doesn't have to squeeze through varying restrictions as the mold fills. When walls aren't uniform, stresses can build at boundary areas between thin and thick walls. This can result in warping, which may compromise the quality and function of the part.

If you absolutely cannot have uniform walls due to the requirements of your design, it is best to gradually transition the wall thickness. Under these circumstances, it may be best to consult an expert with extensive experience in part design for molding.

"IF YOU ABSOLUTELY CANNOT HAVE UNIFORM WALLS DUE TO THE REQUIREMENTS OF YOUR DESIGN, IT IS BEST TO GRADUALLY TRANSITION THE WALL THICKNESS."

Another element to keep in mind is material selection. The following chart displays possible wall thickness selections for different materials. This is yet another important tool in your arsenal as a designer.

ABS0.045 – 0.140	Polyester0.025 – 0.125
ACETAL0.030 – 0.120	Polyethylene0.030 – 0.200
ACRYLIC0.025 – 0.500	Polyphenylene sulfide0.020 – 0.180
Liquid crystal polymer0.030 – 0.120	Polypropylene0.025 – 0.150
Long-fiber reinforced plastics0.075 – 1.000	Polystyrene0.035 – 0.150
Nylon0.030 – 0.115	Polyurethane0.080 - 0.750
Polycarbonate0.040 – 0.150	

Design Considerations

5. SUPPORT/STRAIGHT RIBS THICKNESS

Ribs serve several vital purposes in part design. They strengthen your part's wall sections and bosses while also minimizing the effects of dimensional variation caused by shrinkage. They also help to minimize warp and increase the part's rigidity. They are a uniquely valuable part of preparing your design for plastic injection mold production.

It is important that you balance the thickness of ribs in proportion to the walls. Rib thickness always needs to be less than wall thickness. Generally speaking, rib thickness should be 40-60% of the wall material thickness. If the balance between wall thickness and rib cross section is improper, sinks and distortion can occur causing an undesirable surface finish.

The height of ribs is also important. They should be limited to less than three times the thickness of the material. It is always better to have more short ribs that help increase rigidity than a single tall rib.

Rib orientation is another important aspect to keep in mind. As you design your part for molding, you must also think about its eventual use and bending load. Ribs must be oriented in a direction that will increase stiffness.

Lastly, remember to incorporate draft angles with ribs. A minimum of .25 to .5 on each side is usually sufficient.

Design Considerations

6. MINIMIZE UNDERCUTS AND NEED FOR ACTIONS IN THE TOOL

It is best to eliminate undercuts from your design if at all possible. This is because undercuts prevent your plastic part from being ejected from the mold via normal means, therefore adding a secondary process, which adds time and expense to the process. They also make the mold more expensive to maintain after it is built. Undercuts include threads, grooves, cut outs, protrusions, and snaps.

"YOU NEED TO WORK CLOSELY WITH AN EXPERT IN INJECTION MOLDING TO SEE WHAT IS POSSIBLE."

Obviously, some parts require special elements in order to meet your needs. Just because traditional undercuts aren't advised doesn't mean that molding is impossible. New technology has evolved to assist in molding parts with specific needs in the tool like internal threads, dimples, and even protrusions, but you generally need to work closely with an expert in injection molding to see what is possible.

9360 W. National Road Brookville, Ohio 45309
www.kandbmoldedproducts.com • (937) 833-2178 • Joe.Berberich@KandBMoldedProducts.com

Mold Considerations

7. GATING CONSIDERATIONS

Working with a skilled analyst with plastics processing experience can help you, through mold-filling simulation, to find just the right gate size and location to create the perfect part. Cosmetics can also play a large role in determining gate size and location and should be considered if aesthetics are of high importance for your part.

8. EJECTOR PIN LOCATIONS

Ejector pins assist in removing your part from the core side of the mold. They are an essential part of the molding process. Unfortunately, they can typically leave marks on your part. This is yet another very important element to keep in mind while creating your design. If an outside surface must be impeccable, it is likely better for that surface to be designed on the cavity side of the mold.

Under some circumstances, you may be able to work with the mold builder to designate the location of the ejector pins on the core side of the mold. However, it is always important to be sure the pins are machined at, or slightly below, flush with the core side of the mold. If the pins stick out, the plastic from the part will shrink around them and the part won't release properly. This can lead to a whole host of problems.

Don't fall for the common misconception that waiting longer to open the mold so the plastic is harder and less likely to dent will reduce marks on the part. In truth, the longer you wait, the more the plastic grips to the ejector side of the mold. The pins then have to use even more force to eject the part, which will likely leave the same marks you were trying to avoid in the first place.

Mold Considerations

9. SHRINKAGE

Shrinkage is a natural contraction of the part while cooling in the mold, as well as after injection for several hours, or even several days, while temperature and moisture content stabilize. In addition to designing the structural elements that we have already covered to reduce shrinkage (see: walls and ribs), it is important to keep in mind the material that you are working with. Each material has its own shrinkage rate, which will affect the quality of your product. This means that your choice of material is an important part of the design process.

Before you look at the shrinkage chart below, remember that shrinkage units are expressed as thousandths of an inch per linear inch. The formula is: 0.00X /in/in. Standard shrink rates range from 0.001/in/in to .020/in/in.

ABS:	0.005 - 0.007/in/in	Polyethylene:	0.015 – 0.050/in/in
Acetal:	0.018 - 0.025/in/in	Polypropylene:	0.010 – 0.025/in/in
Acrylic:	0.002 - 0.008/in/in	PP (30% glass):	0.004 – 0.0045/in/in
Nylon 6:	0.006 - 0.014/in/in	Polystyrene:	0.002 – 0.006/in/in
Nylon 66:	0.012 - 0.018/in/in	PS (30% glass):	0.0005 – 0.0010/in/in
Plycarbonate:	0.005-0.007/in/in	PVC:	0.003 – 0.0008/in/in
PET:	0.005 - 0.012/in/in		

Your molder and consultants will also be able to assess and advise you on the best materials for your design.

Glossary Of Injection Molding Terms

Boss: Raised rounded stud feature on plastic parts and molds.

Cavity: The space between both sides of the mold. This is mainly concave and the upper part of the mold, which is filled to create the injection-molded part—sometimes referred to as the "A side" or "A part" of the mold and usually the surface side of the finished product.

Core: Generally the half of the mold where the plastic part is ejected from. Usually at the bottom and sometimes referred to as the "B side" or "B part" of the mold.

Core Outs: Area of the part that is removed to achieve uniform thickness.

Draft: Angle and taper built into walls and ribs of the part that facilitate removal from the mold.

Flash: Excess material that goes outside the intended geometry of the part, usually appearing on parting lines.

Gate: The point where plastic enters the mold through the cavity. There are automatically trimmed gates, which shear when the part is ejected, and manually trimmed gates, which require a secondary operation by an operator in order to be separated from the mold.

Hand Load: A metal feature used to create details like undercuts in molded parts. They are removed manually during or after ejection.

Heel or Lock: This is the piece of a custom automatic injection mold that keeps the slide positioned forward when the mold is closed.

Horn Pin or Cam Pin: Pins that are used to activate the slide on an automatic injection mold.

Line of Draw: The direction in which the two halves of the injection mold separate from the part, thereby allowing it to be ejected without metal obstructions that cause undercuts.

Ribs: Thin features that strengthen wall sections and bosses while minimizing warp.

Runner: A canal cut into custom injection molds, where the plastic travels from the injection-molding machine, through the sprue, runner, and the gate until it eventually fills the mold to make the part.

Shear: The force created between layers of resin as they glide against one another during various phases of the injection molding process. This can cause heat to develop.

Short Shot: When a plastic part doesn't fill the mold completely. This results in features being absent from the part.

Shrink Rate: The rate of how much plastic will shrink when cooled, ranging from .001 per inch to as much as .060 per inch. Most shrinkage rates fall in between .004" and .021."

Side Action: Phrase used to describe the hand pulls or slides commonly utilized in injection molding.

Sink Marks: Depressed areas of a molded part due to thickness ratios being off in walls and ribs.

Slide: Feature required in automatic injection molds for creating undercuts.

Sprue: The canal that connects the injection molding machine nozzle to the runner.

Steel Safe: The amount of metal left on the mold in order to fine-tune a dimension. An example would be if you have an inside diameter that needs to be .300" you may leave the mold at .305: in case of shrinkage.

Thin Wall Molding: When molded parts have walls between .005" to .060" thick.

Undercuts: Area of the designed component where a slide or hand pull is necessary to make clips, windows, or holes that are not in the line of draw.

Vestige: Excess material that protrudes from the gate and usually needs to be trimmed by the molding machine operator after the gate runner has been removed from the injection molded part.

Wall Thickness: The thickness of a cross section in a plastic part.

Warp: A distorted area of an injection molded part. This usually happens after molding or during cooling and is often caused by residual stresses in the part resulting in differential shrinkage.