

# **How to Avoid the Common Pitfalls of Plastic Product Design**

## **The Five Basic Rules**



**United Plastics Group, Inc.**

# How to Avoid the Common Pitfalls of Plastic Product Design

## The Five Basic Rules

As in life there are some basic rules in plastics that if followed, will ease the entire process. Actual plastic product design for injection molding is simple if you follow these basic rules.

Plastics are a fluid ruled by fluid dynamics. If you studied fluid dynamics, you will remember that as a fluid flows through a pipe it experiences pressure changes as it travels from one diameter pipe to another. When you go from a small pipe to a large pipe there is a pressure drop. When the fluid goes from a large pipe into a small pipe the fluid backs up and the pressure increases to push the fluid from the bigger pipe into the smaller pipe. Plastics are fluids. When plastic flows from a thin section into a thick section there is a pressure drop. When the plastic is going from a thick section to a thin section the pressure increases.

Changes in pressure, velocity, and plastic viscosity, especially as the plastic cools, all result in a variety of manufacturing issues. Keeping dramatic changes in plastic part design to a minimum will obviously keep manufacturing issues to a minimum. The following few rules will assist product designers in avoiding the common pitfalls.

### Rule 1, Uniform Wall Thickness

As a designer, you can't spend your entire day figuring out the fluid dynamics of plastic flow while you are designing--you'll never complete the design. So keep it simple. Keep the walls of the plastic part uniform and you will not need to hear that

the part you designed warped, could not be filled or the parts don't fit together because the part shrinkage is strange.

It might sound as if we are simplifying a very complex science of fluid dynamics and stress. In fact, that is exactly what we should be doing in plastic part design. As soon as you violate the most basic design parameter of plastic design -- uniform wall thickness -- you are headed for trouble. With high mold shrinkage plastics, a designer should try to limit wall thickness variations to 10 percent. A 10 percent variation in thickness will be enough to cause processing and quality issues.

Keep the walls of a plastic part uniform. More problems with part warpage, varying shrinkage and part fill are the result of varying wall thickness. If you follow this most basic rule of plastics, you will not face many of the issues associated with manufacturability of plastic parts.

### Rule 2, Gate Location

Faced with the need to vary wall thickness, the designer must provide for proper gate location. Gate location is best if the melt enters the cavity in the thickest area and flows to the thinner areas. If this is not done, it will be virtually impossible for the uniform pack out of the molded part.

### Rule 3, Nominal Wall Thickness

Nominal wall thickness is another design rule that will keep you out of trouble. Injection molding is a melt flow process that has limitations on how far the molten plastics will flow. Theoretically, there is no limit to maximum wall thickness that

can be injection molded. There is, however, a definite limit to how thin a wall can be.

The criteria for selecting a wall thickness have to be based on the requirements of the product. Strength determines the wall thickness. From a functional perspective, interpretation of a finite analysis can be relied upon to optimize selection of suitable wall thickness.

The cost of a molded part is dictated by its size, shape, and the plastics specified. For a given plastic material, a part's cost relates directly to thickness. Thicker parts require more plastic material. Thick-walled plastic parts require longer cooling cycles. These two cost factors contribute significantly to the cost of an injection molded plastic part. Thinner is always better in unfilled plastics parts (which are those parts that don't require glass or carbon fiber reinforcement)

If a designer is faced with a filled plastic (one that requires reinforcement by glass or carbon fiber) fiber alignment can be an issue. Wall thicknesses below 2 mm do require fiber alignment review.

#### **Rule 4, Radius Corners**

Again, injection molding is a melt flow process. Let's go back to our fluid dynamic course again. When a fluid flows through a maze and reaches a turn or corner, it needs to go around the corner. Rounded corners improve the flow of the plastics. So, the fourth rule is to radius the corner of all your parts generously.

The injection molding process is unforgiving of sharp inside corners. During the cooling portion of the molding cycle, the top of the part attempts to become small-

er. The steel core controls the inside diameter. This cooling process creates a high level of molded-in stress, as the material is "stress" against a sharp corner.

If you design a part so the inside radius is half the nominal wall and the outside radius one-half times the nominal wall, the proportions produce a uniform thickness around the corner. Shrinkage will be the same on both sides of the corner and there will be no sink marks. And, by doing this you just maintained the first golden rule of plastics, uniform wall thickness.

As the melt flows around a properly proportioned corner there is no increase in area and no abrupt change in direction. If you follow this rule, there is minimum loss in cavity packing pressure which will result in a good, strong corner with a reduced tendency for post mold warpage. A corner of this type will be dimensionally stable.

#### **Rule 5, Draft Angles**

What goes in must come out, we hope. The justification for draft angles comes from the nature of the injection molding process and the ever-present nemesis of mold shrinkage. Injection molding is a high-pressure process. These high pressures force the plastics into intimate contact with all surfaces of a mold's cores and cavities. This high-pressure packing of the cavity makes it difficult to demold a part.

In some cases, shrinkage of the plastic material will pull the part away from the cavity, making the demolding easier. In other instances, shrinkage causes the plastic material to tightly grip the cores that form the inside surfaces of the part. This natural occurrence requires draft angles to remove the plastic part without distortion

or damage.

There is no single draft angle that is adequate for all injection molded parts. Each individual part has its own unique draft requirements. Large parts require more draft than small parts. Thin-walled parts that are molded at high pressure require more draft than parts molded at lower injection pressures.

As the plastic material cools, it shrinks and pulls away from cavities while gripping the core pins. Theoretically, core pins require more draft than cavities. The amount that a plastic material shrinks must be considered in selecting suitable draft angles.

Large draft angles and smooth polish are required for parts molded in strong, brittle, abrasive and sticky materials. Smaller draft angles can be used on soft, ductile, and slippery materials.

The ideal draft angle, from a cost and manufacturability perspective, is the largest angle that will not distract from the customer's acceptance of the product. The minimum allowable draft angle is more difficult to define. The hands-on experience gained by plastic material suppliers and molders makes them the ultimate expert on the minimal acceptable draft.

In most cases,  $1^\circ$  per side will be adequate, but  $2^\circ$  and  $5^\circ$  per side would be better. If the design cannot tolerate  $1^\circ$ , then specify  $1/2^\circ$  per side. A minimal draft angle, such as  $1/4^\circ$  or even  $.002$  inch/side, is better than no draft angle at all.

When designing a part, the engineer may not know the location of the mold's parting line. Without that information, it is impossible to determine whether the part

should have plus or minus draft angles. There is also confusion as to how much is required. This lack of knowledge has resulted in the common practice of drawing the part without draft angles and specifying the drawing as a drawing note, such as "allowable draft  $1^\circ$ ". This technique simplifies the part design process. It is an undesirable draft angle specification that leads to misinterpretation.

Experienced designers are aware of the importance of molding draft angles. Most designers try to specify draft angles on the sidewalls of a part that are perpendicular to the mold's parting line. Many designers will, however, overlook drafting the other details of the part. Draft angles on mounting flanges, gusset, holes, hollow bosses, lovers, and other holes will all benefit plastic parts.

Any draft angle is better than none. The important cost savings benefits of draft angles will, however, be lost by a half-done job. Failing to provide a draft angle on a long side-acting core pin that forms the hole in a threaded projection would be a serious mistake. The force required to pull that core pin could be one thing that requires a longer cooling cycle.

### **Many More Design Criteria**

There are many more items to be discussed in plastic product design, such as rib design, weld lines, plastics thread design, inserts and so on. There are thick and thin books on plastics design which cover in great depth the many other technical issues that need review. All of these are important considerations in designing your plastic parts, however, the five basic rules noted are the foundation for all good plastic medical product design. If one violates the basic five rules, the other design requirements will be compromised.

Take a piece of paper and write down the following, or else cut out this section of the article and tape it to your CAD station:

### **Good Plastic Design Practices**

- 1) Keep Wall Thickness Uniform**
- 2) Consider Gate Location, Thick to Thin Section Fill**
- 3) Keep Nominal Wall Thickness to a Minimum**
- 4) Radius Corners**
- 5) Draft Angles, More is Never Bad**
- 6) When all else fails call Bob Alvarez at United Plastics Group**

For More Information, please contact:

Bob Alvarez  
Vice President, Technology  
United Plastics Group, Inc.  
3125 E. Coronado St.  
Anaheim, CA 92806  
714-575-5384  
balvarez@upgintl.com

