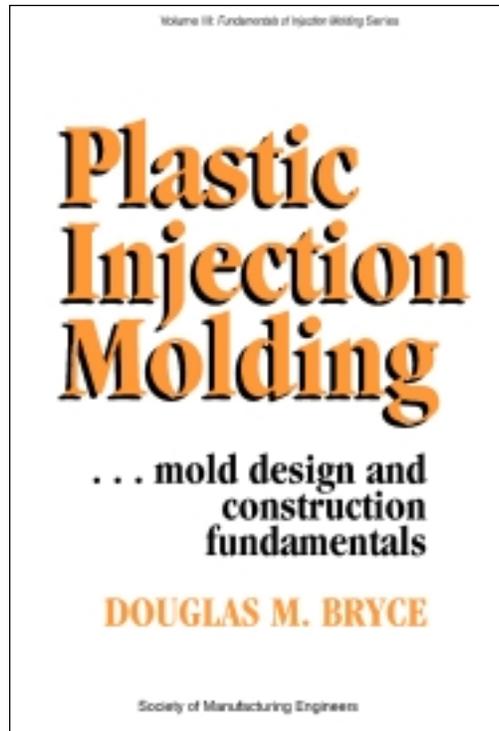


This PDF document is a
partial sample chapter
from the book...



To order this book,
print the last page of this
document.

Copyright ©1998



Society of
Manufacturing
Engineers

GATHERING INFORMATION

Before a mold design is started, there are some basic facts that need to be gathered. These include determining how many cavities to build, what material to make the mold out of, and other data that we will discuss in this section. It is probable that more than one area of expertise may be required to obtain all the information that is necessary. This may result in soliciting specialized assistance from others such as material engineers and financial analysts. However, we can make some basic assumptions, based on whatever knowledge and information is available at the time.

Which Plastic?

While the mold designer usually does not select the molding material, he or she should be aware of the more important aspects and characteristics involved in molding specific plastics. For example, shrinkage factors sometimes vary widely between different materials and may vary among different grades and versions of the same material. Also, some plastics will absorb and dissipate heat more efficiently than others, resulting in more efficient cooling during the molding process. This may affect cooling channel locations in the mold, and the viscosity of a particular plastic has a large influence on gate, runner, and vent design, location, and construction.

A thorough study of the characteristics of various plastics is not possible within the scope of this book, and the reader is advised to locate such information in Volume 2 of this series: *Material Selection and Product Design Fundamentals*. Instead, we will discuss the basic information that is desirable to know about a specific material as it applies to mold design and construction.

Determining Shrinkage

Every material we know of (except water) expands when it is heated and contracts when it is cooled. In the field of plastics, we define the contraction phase as *shrinkage*. Each plastic material has a *shrinkage factor* (sometimes incorrectly called a shrink rate) assigned to it. This factor is used to estimate how much a part will shrink after it is removed from the mold. After that is determined, the mold

can be built to a set of dimensions that create a molded part large enough so that it will contract to the desired finished size after shrinkage.

Those plastics that shrink equally in all directions (notably amorphous materials) are referred to as having *isotropic shrinkage*. Some plastics (notably crystalline materials) will shrink more in the direction of flow than across the direction of flow (unless they are reinforced, in which case, shrinkage is greater across the direction of flow). This type of shrinkage, which is not equal in all directions, is known as *anisotropic shrinkage*. Both types of shrinkage are shown in Figure 2-1.

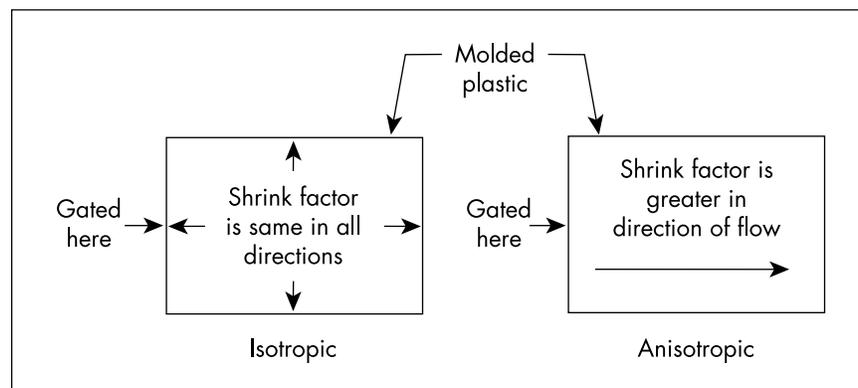


Figure 2-1. Part shrinkage.

Shrink factors are assigned on an inch-per-inch basis, meaning that the factor is applied to every inch (or fraction of inch) of every dimension of the product to be molded. For example, if a product is to be 6 in. (15.2 cm) in length, and the shrink factor is .010 in./in. (0.010 cm/cm), the mold cavity must be made to be 6.060 in. (15.4 cm) in length to produce the 6-in. (15-cm) long product.

The shrink factors are rated as low, medium, or high. *Low shrink* is commonly accepted as from .000 in./in. to .005 in./in. (0.000 cm/cm to 0.005 cm/cm). *Medium shrink* is commonly accepted as from .006 in./in. to .010 in./in. (0.006 cm/cm to 0.010 cm/cm), and *high shrink* is commonly accepted as anything over .010 in./in. (0.010 cm/cm). Some plastics may have as great as .075 in./in. (0.075 cm/cm) shrinkage. Amorphous materials tend to exhibit low shrink, semicrystalline materials have medium shrink, and crystalline materials tend to show high shrinkage. If glass reinforcement is added to the plastic, the shrinkage will be less than that same plastic in a *neat* (no reinforcement added) condition. That's because the nonshrinking glass reinforcement takes up some of the volume of the mass of plastic and dilutes the shrinkage of the total mass.

Shrinkage is difficult to estimate because there are many items that influence the final shrinkage result. Changes in wall thickness of the product design may cause different shrinkage to occur in certain areas of the molded part. Temperature variations in the mold (greater than 10° F [5.5° C] between any two points) may result in varying shrinkage areas of the molded part. The mold designer can only use a best guess method of determining shrinkage by gathering as much advice from experienced molders, moldmakers, material suppliers, and other designers as possible, and then staying steel-safe on all dimensions to allow adjustments after sampling the mold.

It is not unusual to have to *develop* the mold according to the shrinkage idiosyncrasies of the molding material. For example, in a case where a hole is molded by using a core pin fastened in the mold, the core pin will be made to the part dimension diameter plus shrinkage. But, the finished product may be molded with the hole having an elongated diameter rather than a perfectly round one, due to shrinkage conditions. After the molder has successfully optimized the molding process, the core pin may have to be machined to be in an out-of-round condition so the final molded part, after cooling and shrinking, will actually have a truly round hole diameter. This development process might occur in other areas of the cavity because of the molding material's shrinkage characteristics and the product design.

Pressure and Viscosity

Viscosity is a measurement of the thickness of a material in its liquid (molten) state. The higher the viscosity, the thicker the material. A high viscosity plastic material requires a greater amount of injection pressure to push it through the mold than a low viscosity plastic. In addition, the high viscosity materials require larger runner diameters and greater gate volume to allow easy flow to the cavity image. And, the high viscosity plastics allow deeper vents for faster removal of trapped air.

The viscosity of a plastic determines how much pressure will be needed to inject the material into a mold. Viscosity is measured by elaborate and relatively expensive test equipment. But, it can be indicated inexpensively by using an ASTM test D1238, which uses a small amount of plastic material and simulates the injection molding process, as shown in Figure 2-2. This test is called the *melt index* test, but is also called *melt flow*, *flow index*, and *melt rate*. A machine called a *plastometer* is programmed to a set of conditions dictated by the plastic being analyzed. Currently, there are at least 33 sets of conditions and each plastic will fall within one of these conditions.

The melt index number can be used as a tool for determining the flowability of a particular plastic. The test begins by dropping an amount of raw plastic in the heating chamber, placing a plunger device in the chamber, setting a predeter-

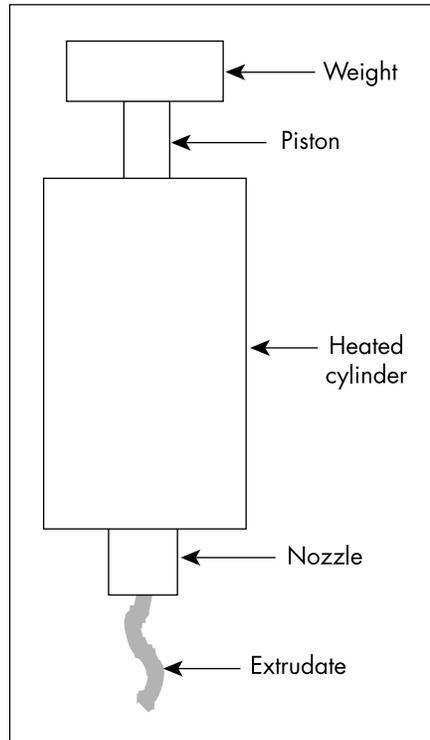


Figure 2-2. Melt index test apparatus (viscosity indicator).

mined load on top of the plunger, and measuring the amount of material that extrudes from the nozzle in a 10-minute period. The resultant number is the melt index value measured as grams per 10 minutes. Flow numbers (MI numbers) usually fall within a range of 2 to 50 with 12 to 14 being the most common (roughly half an ounce per 10 minutes). This means that 12 to 14 grams of plastic came from the orifice of the test machine in a total of 10 minutes.

The lower the melt index number, the stiffer the flow of the material. That means a higher injection pressure will be required to fill the mold, larger runner diameters will be needed for surface runner molds, and gate depths will probably increase. Vent depths are affected by viscosity; as the MI number gets smaller, the vents are made deeper to allow trapped air to escape the mold faster and reduce injection pressure requirements.

Viscosity also affects physical properties of the molded part. Basically, the higher the MI number within the range

associated with a given plastic, the weaker the molded part. Conversely, the lower the MI, the stronger the part. Some of the ways that molded part properties are affected by viscosity are shown in Table II-1.

How Many Cavities?

Before we can determine the size of mold and the size of equipment needed to run the mold, we must determine how many cavities are required. Along with the total time of a cycle, the number of cavities determines how many molded parts can be produced during one complete cycle of the injection molding process. The number of cavities needed depends on the time frame established for producing the annual volume requirements of a specific product. For example, if an average of 100,000 units a year is required, we need to determine how many cavities are required to produce the product during the year. First, determine the production time available for the year. Most molding operations produce parts 24 hours a

**Table II-1. Effect of Viscosity on Physical Properties
(Relationship as Melt Index Value Decreases)**

Stiffness	Increases
Tensile strength	Increases
Yield strength	Increases
Hardness	Increases
Creep resistance	Increases
Toughness	Increases
Softening temperature	Increases
Stress-crack resistance	Increases
Chemical resistance	Increases
Molecular weight	Increases
Permeability	Decreases
Gloss	Decreases

Note: Permeability and gloss actually decrease as the melt index value drops.

day, five days a week. Weekends are used for maintenance. Assuming a 52-week year, 5 days a week, and 24 hours a day, we arrive at a total time of 6,240 hours a year. Each month then has an average of 520 hours available (6,240/12).

To calculate how many cavities we will need to machine into the mold, we will have to estimate a cycle time. The cycle time is determined primarily by the thickest wall section of the part. For a guideline, Figure 2-3 can be used to make this determination and assumes that the mold will be placed in a properly sized molding machine and that all phases of the injection process are average times. Different materials may result in longer or shorter times, but Figure 2-3 is the result of actual tests performed by Texas Plastic Technologies from 1991 through 1994.

Note that the chart line in Figure 2-3 does not rise at a straight angle. That is due to the cooling time portion of the overall cycle. As the wall thickness doubles, the cooling time actually increases four-fold. That is why it is beneficial to keep wall thicknesses (and gate thickness) at an absolute minimum.

After the total cycle time is estimated, using Figure 2-3, the number of cycles per hour can be calculated by dividing 3,600 (the number of seconds in an hour) by the estimated cycle time. Let's make an assumption that the part in question has a maximum wall thickness of .100 in. (25.4 mm). From Figure 2-3, we find that the total cycle time would be approximately 36 seconds. Dividing that number into 3,600 shows that we can mold 100 cycles per hour. Now, we can figure out how many cavities we need. If we have a single cavity mold we can produce 100 units per hour. That means it would take approximately 1,000 hours, or 8.33 weeks, to mold our annual requirement of 100,000 units. If we built a two-cavity mold we