MOLD DESIGN BASICS
Part One

GATHERING INFORMATION

Before any work is actually done to create a mold design, 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 a variety of 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 assistance from others trained in the respective areas of concern such as Material Engineers, and Financial Analysts. But, we can make some basic assumptions, based on whatever knowledge and information is available at the time, by utilizing the methods outlined below.

WHICH PLASTIC?

While the mold designer is not normally the one to select which plastic material is to be molded, he/she should be aware of some of the more important aspects and characteristics involved in molding specific plastics. For instance, shrinkage factors sometimes vary widely between different materials and may vary even 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 effect cooling channel locations in the mold. And the viscosity of a particular plastic has a large bearing 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, titled ***Material Selection And Product Design*** . Instead, here 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. Once 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.

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. Those plastics that shrink equally in all directions (notably amorphous materials) are referred to as having isotropic shrinkage. Both types of shrinkage are depicted in the following sketch:

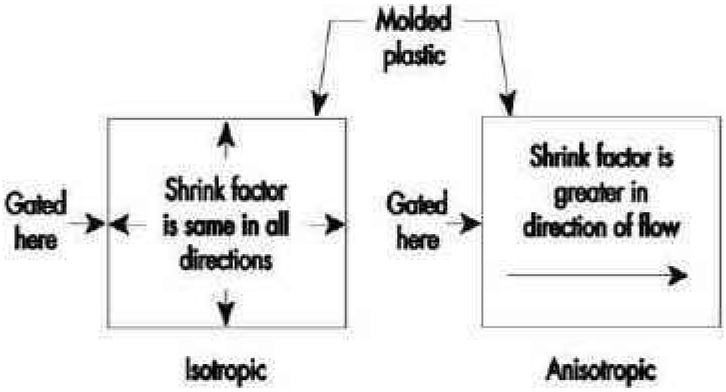

Figure 1-1 Isotropic And Anisotropic Shrinkage

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

The shrink factors are rated as low, medium, or high. Low shrink is commonly accepted as from 0.000 in/in to 0.005 in/in. Medium shrink is commonly accepted as from 0.006 in/in to 0.010 in/in, and high shrink is commonly accepted as anything over 0.010 in/in. Some plastics may have as great as 0.075 in/in shrinkage. Amorphous materials tend to exhibit low shrink, semi-crystalline 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 non-shrinking glass reinforcement takes up some of the volume of the mass of plastic and dilutes the shrinkage of the total mass.

Shrinkage is impossible to predict and a difficult thing to estimate due to the number of 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 degrees F between any two points) may result in varying shrinkage results across a certain area 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, mold-makers, 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 due to the idiosyncrasies of the shrinkage phenomenon. An example of this might be in a case where a hole is molded into the product by using a core pin fastened in the mold. This 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. Once 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 "developing" might need to take place in other areas of the cavity also, due to the shrinkage characteristics of the plastic material that was selected, and the product design that is being followed.

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 way of elaborate and relatively expensive test equipment. But, it can be indicated inexpensively by using an ASTM test #D 1238 which uses a small amount of plastic material and simulates the injection molding process. This test is called the Melt Index test, but also goes by other common names including Melt Flow, Flow Index, and Melt Rate. A machine called a ***plastometer*** is programmed to a set of conditions dictated by which plastic is being analyzed. There are at least 33 sets of conditions as of this writing and each plastic will fall within one of these conditions.

This 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 a desktop heating chamber, placing a plunger device in the chamber, setting a pre-determined 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. 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 higher injection pressure will be required to fill the mold, and larger runner diameters will be needed for surface runner molds. In addition, gate depths will probably need increasing. And, vent depths are affected by viscosity; the lower the MI number, the deeper the vents can be, thus allowing trapped air to escape the mold faster and reducing 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 the following chart.

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 that Permeability and Gloss actually *decrease* as the Melt Index value drops.

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 instance, if we have an estimate that we will require an average of 100,000 units a year we wish to determine how many cavities we need to produce them during the year. Most molding operations produce parts 24 hours a day, 5 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 amount of available time to be 6,240 hours a year. Each month then has 520 hours available, on average (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, the following chart (Figure 1-2) can be used to make this determination. The chart assumes that the mold will be placed in a properly sized molding machine and that all phases of the injection process are timed on the average. Different materials may result in longer or shorter times, but this chart does represent actual tests performed (by Texas Plastic Technologies from 1991 through 2002) on a variety of materials, and the results have been averaged.

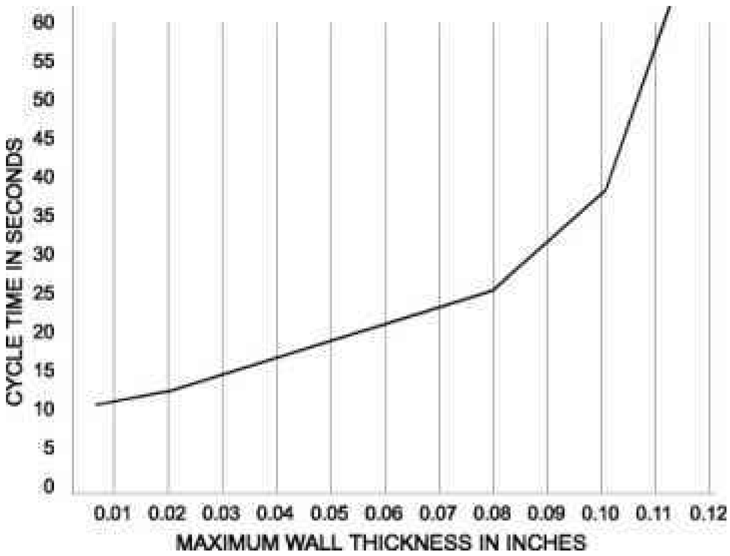

Figure 1-2 Estimating Cycle Times

Note that the chart line 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.

Once the total cycle time is estimated, using the above chart, the number of cycles per hour can be calculated by dividing 3600 (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 0.100 inches. From the chart we find that the total cycle time would be approximately 36 seconds. Dividing that number into 3600 shows that we can mold 100 cycles per hour. 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 requirements of 100,000 units. If we built a 2 cavity mold we could mold the total 100,000 units in half the time, or 4.16 weeks. Of course, the two cavity mold would be more expensive and this must be considered.

Now let's consider a situation where we need *10 million* units a year. If the cycle time stays at 36 seconds we still have a total of 100 cycles per hour. Therefore a single cavity mold could make as many as 624,000 units if it ran all year long. We would need at least a 16 cavity mold to produce all 10 million parts on one mold and that would mean running the mold constantly, even a few overtime hours on weekends. This is not normally practical. What we might consider is 2 to 4 molds, each with 16 to 32 cavities each. Then we could produce the entire year's worth of requirements in 3 to 6 months molding time, depending on which combination of cavity and mold numbers we decided to use. Also, we would be utilizing 2 to 4 molding machines at once and that must be taken into consideration, especially in a proprietary molding facility. If we do not have enough molding equipment on hand (or the proper size) we may have to farm out the molding, or we could purchase additional equipment if it can be justified. These are just some of the typical issues that must be addressed before a determination can be made as to how many cavities should be placed in a mold.

WHICH MOLD MATERIAL?

There are dozens of materials that can be used for making molds for producing plastic products, including many types of steels, aluminum, brass and copper, epoxy, and many others as well as combinations of these. Following are some of the more common materials and the role they play in the making of molds.

STEELS

1020 Carbon Steel - Used for ejector plates and ejector retainer plates. Easily machined and welded. Can be hardened to Rc 20-30.

1030 Carbon Steel - Used for mold bases, ejector housings, and clamp plates. Has 25% greater tensile strength than 1020. Easily machined and welded. Can be hardened to Rc 20-30.

1040 Carbon Steel - Commonly used for support pillars. Tough steel with good compressive strength. Can be hardened to Rc 20-25.

4130 Alloy Steel - This is a high strength steel used primarily for cavity and core retainer plates, support plates, and clamping plates. Supplied at 26-35 Rc.

6145 Alloy Steel - Primary use is for sprue bushings. Supplied at 42- 48 Rc.

S-7 Tool Steel - Shock resistant with good wear resistance. Used for interlocks and latches. Hardened to 55-58 Rc.

O-1 Tool Steel - General purpose oil-hardening steel. Used for small inserts and cores. Hardened to 56-62 Rc.

A-2 Alloy Tool Steel - Good dimensional stability and abrasion resistance. Used for hobs, slides. Hardened to 55- 58 Rc.

A-6 Tool Steel - Good dimensional stability, high hardness, general purpose oil-hardening steel. Primary use is for optical quality cavities and cores. Hardened to 56-60 Rc.

D-2 Tool Steel - High chromium, high carbon content. Difficult to grind, but excellent abrasion resistance. Used for gate inserts, lifters, slides. Hardened to 58- 60 Rc.

H-13 Tool Steel - Very high toughness, low hardness steel. Used for high quality cavity and core requirements. Also primarily used foe ejector pins, return pins, sprue pullers, leader pins, and slide actuating angle pins. Supplied annealed at 15- 20 Rc but can be hardened to 60 Rc with little distortion.

P-20 Tool Steel - This is a modified 4130, commonly referred to as "pre-hard". It is supplied at a Rc hardness of 28- 40 which provides moderately high hardness, good machinability, and exceptional polishability. It is used primarily for cavities and cores as well as stripper plates.

420 Stainless Steel - Used in applications requiring exceptional chemical resistance (such as when molding PVC resins), this steel is usually supplied in an annealed condition (15- 25 Rc) but can be hardened to 55- 60 Rc. Its primary use is as a steel for cores and cavities.

ALUMINUM

While there are many grades of aluminum available for making molds, the most common, and most efficient to work with, is the aircraft grade 7075 (T6). This wrought aluminum alloy is produced by hot rolling cast aluminum to the desired thickness of plate to be used. The entire mold can be made of the same material (including cavity and core) and an anodizing process can be utilized to impart a surface hardness of up to 65 Rc for wear resistance. However, due to the "smoothing" tendency of the normal aluminum surface, it is possible to mold with no surface treatment. The microscopic "hills" and "valleys" of the aluminum surface tend to even themselves out without galling. Use of 7075 aluminum can result in mold build times being reduced by up to 50% (due to faster machining times) and injection molding cycle reductions of up to 40% (due to faster heat dissipation) depending on size and complexity of the product being molded.

Until recently, aluminum has been considered as a mold making material only for low volume production or prototype molds. The use of 7075 alloy has created opportunities to use aluminum for high volume production up to millions of cycles. Even glass-reinforced and high temperature plastics can be molded successfully in aluminum molds.

BERYLLIUM-COPPER ALLOYS

High strength and high levels of thermal conductivity make the beryllium-copper alloys excellent selections for making cores and cavities for injection molds. They are usually used as components that are fit to steel mold bases, but can also be used in conjunction with aluminum mold bases for greater economy. They find a special use for situations where the placement of cooling channels in the mold makes removal of heat difficult, such as in deep draw parts or parts with unusual contours. Strategically placed beryllium-copper components will assist in dissipating the heat from these areas without using complicated water line channels.

The types of beryllium-copper most commonly used for cores and cavities are CuBe 10, CeBu 20, and CeBu 275. They differ mainly in tensile strength with the higher numbers having the greater strength. In addition, the higher number grades allow higher levels of hardness. This ranges from a low of Rb 40 for CeBu 10 to a maximum of Rc 46 for the CeBu 275.

(It must be noted that beryllium-copper dust is carcinogenic, and care must be exercised when machining it, to avoid breathing any dust that might be generated).

OTHER MATERIALS

There are other materials that can be used for making molds for plastic injection molding, including epoxy, aluminum/epoxy alloys, silicone rubbers, and even wood. However, these are usually all reserved for very small volumes such as only 1 to 12 pieces. In most cases, these represent prototype volumes, and the molds are not expected to meet the demanding requirements of higher volume production levels. The scope of this type mold construction does not fit the intent of this publication, and the reader requiring further information is encouraged to contact mold makers specializing in the prototype field. A listing of these can be obtained from the Society of the Plastics Industry (SPI), 14 Fairfield Dr., Brookfield, CT., 06804-0403.

End Of Part One