CARL HANSER VERLAG

Herbert Rees

# Mold Engineering

2nd edition

3-446-21659-6

www.hanser.de

# 1 Introduction to Mold Engineering

*Mold Engineering* deals with injection molds for thermoplastic molding materials. However, much of the subjects treated herein apply also to other molds, such as blow molds, and to some types of thermoset molds.

The purpose of Part I: Mold Engineering is to familiarize not only beginners in mold design but also more experienced mold designers with an engineering approach to mold design. *Mold Engineering* is also recommended literature to any individuals, such as sales and service personnel, in the plastics industry who need to understand molds in the course of their activities. Part I: Mold Engineering is divided into three sections:

Section 1: Basics About Molds, Machines, Plastics, and Products,
Section 2: General Mold Design Guidelines, and
Section 3: Specific Subjects for the Mold Designer.

There are several other good books on mold design. Some of them are listed at the end of this chapter [1–4]. These books contain designs and illustrations of molds built for very specific applications and plastics. They show many complete mold assembly drawings having many different features, without explaining in detail *why* these features were selected. In many cases, the molding technology shown is about 20–30 years behind current standards. This is not to say that these books are without value. On the contrary, a designer may find many interesting solutions to problems that may arise when designing a mold, provided one does not just copy them but uses ideas from these designs in combination with today's technology.

Part I: Mold Engineering, rather than showing complete assemblies, breaks the mold design down into its various features (elements) and treats them separately, from an engineering point of view. It also explains *why* and *when* certain features are to be selected, and *when not*.

## 1.1 What is an Injection Mold?

An injection mold is a arrangement, in one assembly, of one (or a number of) hollow *cavity spaces* built to the shape of the desired product, with the purpose of producing (usually large numbers of) plastic parts, or *products*.

The cavity space is generated by a female mold part, called the *cavity*, and a male mold part, called the *core*. To fill the cavity spaces, the mold is mounted in an injection molding machine that is timed (usually automatically) to:

- *Close* the mold,
- *inject* the (hot, more or less fluid) plastic into the cavity spaces,

- *keep the mold closed* until the plastic is cooled and ready for ejection,
- *open* the mold, and
- *eject* the finished products. Also,
- if necessary, the machine may stay open an additional *mold open (MO) time* to ensure that the mold is ready for the next (injection) cycle, before closing.

The *molding cycle* (in seconds) in a fully automatic (FA) operation is defined as the time from the moment the mold is closed for one injection, or *shot*, until it is closed again for the following shot. Usually, the number of shots per minute (or shots per hour) are given to indicate productivity of a mold, rather than the molding cycle in seconds.

## 1.2 What is an Injection Molding Machine?

It is important that the designer first understands the action and terminology of the injection molding process. An injection molding machine consists essentially of four (4) distinct elements:

1. clamping mechanism,
2. plasticizing unit,
3. injection unit, and
4. all necessary controls.

### 1.2.1 Clamping Mechanism

The clamping mechanism opens and closes the mold (preferably rapidly) as required during the cycle. It must also supply the necessary *clamping force* to keep the mold closed during injection, because the injection pressure acting on the internal, or *projected*, surface of the cavity space tends to open the mold at the *split-* or *parting-plane*, also called *parting line (P/L)*.

### 1.2.2 Plasticizing Unit

Today's plasticizing unit is almost exclusively an *extruder* that heats the cold plastic material to the required temperature to make it fluid for injection, or *melt*. The heating is generated mostly by the mechanical energy (created by the screw motor) as the extruder screw rotates in the barrel and works the plastic. This screw action also advances the plastic toward the tip of the screw.

Heaters around the barrel, usually in three or more heating zones, provide additional heating, which is mainly required during start-up of the machine but also where the mechanical working of the screw alone would not plasticize the amount of plastic required for each shot.

## 1.2.3 Injection Unit

An injection unit forces the melt (under pressure) into the mold. The level of pressure required to fill a mold depends largely on the wall thickness of the product.

Injection pressure is defined as the pressure in the plastic at the point where it enters the machine nozzle. Heavy-walled products require relatively low pressure (50 to 100 MPa, or 7,000–14,000 psi). There are even cases where the extruder pressure alone is sufficient to fill a cavity (flow molding). Thin-walled products, especially if the *L/t* ratio (see definition on page 7) is greater than 200, may require much higher injection pressures (even twice as much) to ensure that the cavities will fill before the plastic freezes.

There are two injection methods used, either single-stage or two-stage, and the types of machine used are discussed below.

### *1.2.3.1 RS Machines*

Today, in most injection molding machines, the extruder and the injection unit are combined into one unit. The extruder screw is stopped when enough melt is prepared for the next injection, then the screw is pushed forward to inject the melt accumulated in front of the screw tip. These units are called by various descriptions: in-line, ram screw or reciprocating screw (RS), or single-stage injection units.

Manufacturers rate screws by the amount of plastic they can plasticize per hour. However, due to limits imposed by the thrust bearing size and strength, the extruder can only plasticize during that portion of the molding cycle when it is *not* injecting; therefore, it is actually used at less than its “rated” plasticizing capacity (indicated in the machine specifications). With large shots, and with slow injection, the time required to inject takes a relatively large portion of the molding cycle, and the screw is often able to plasticize, or *recover*, only 60–80% of its rated capacity.

The amount of plastic pushed into the mold depends also to a large extent on the efficiency (tightness) of the *check valve* at the tip of the screw. If this valve is poorly designed or worn, some plastic will leak past or through the check valve during the high pressure injection cycle. This will affect the amount of plastic entering the cavity spaces, and thereby the productivity of the mold, by causing *short shots* or *packing* (over-filling); this in turn affects the mass (density), size (due to varying shrinkage) and, in general, the quality of the product.

To avoid some of the effects of varying shot size, molders using RS machines usually prepare a shot size greater than required for the shot, so that the screw will never come completely forward, or *bottom out*, but create a *cushion* of melt of about 5–10 mm (0.25–0.50 in.) at the tip of the screw.

### *1.2.3.2 P Machines*

A preplasticizing machine system separates the functions of the extruder and the injection unit. The extruder plasticizes the material and fills an injection cylinder, or *pot*, of the injection unit. These machines may be called preplasticizing, two-stage, or simply P machines.

Advantages of the two-stage system are:

- The screw can run continuously, therefore plasticizing 100% of the available time (and its rating). This may permit the use of a smaller extruder than for an equivalent size RS machine.
- By running continuously, the P machine produces melt that is better mixed and can be held at lower temperatures than with an RS machine. This may be very desirable with certain heat-sensitive materials.
- There is no check valve at the screw. Also, the shot volume in the injection pot is mechanically measured, and the repeatability and accuracy of the shot size is greater than with an RS machine. No cushion is required, and the volume of plastic injected can be matched perfectly to the volume of the cavity space(s).
- Because the transfer from extruder to pot takes place under very low pressure, it is easily possible to place an effective *filter* in the path of the plastic to remove any dirt in the plastic. This will not affect (reduce) the injection pressure from the pot to the mold. Such filtering is not practical with RS machines,where too much pressure would be lost in a filter.

Disadvantages of this system are:

- Higher cost of the machine, because more hardware and more controls are required, and
- this system is not suitable for some very heat-sensitive materials, such as PVC.

### 1.2.4 Controls

Controls make the molding machine operate. There are four basic elements of molding machine controls:

- The *command module* is located near the safety gate of the clamp, where the operator can observe the mold. There, the operator has easy access to the pushbuttons to operate all functions manually. In some machines, the operating push buttons and the controls for the machine settings are in the panel near the safety gate.
- The *control logic* executes the machine settings and manipulates the signals from positional sensors, timers, etc., to make the machine perform as specified. Today, the machine logic operates almost exclusively using electronic switching or uses a microcomputer.

  [Note that mechanical relays, limits switches, and timers have a much shorter service life than electronic switches or timers, and are much less reliable and repetitive. However, they are easier to understand and to service than electronic switching, which requires better qualified service personnel and better electrical measuring instruments for checking and servicing. On the other hand, electronics are more sensitive to elevated temperatures found in some hot countries and may require provisions for cooling the control cabinet.]
- The *power supply* and distribution to the motors and heaters, and
- The *heat controls* for the machine and mold heaters.

There are other features of an injection molding machine, mostly for the convenience of the user. However, for the purpose of understanding the injection process, the above described basic elements are sufficient.

## 1.3 Mold Timing and Terminology

*Dry-cycle:* The *total* time required for the (machine) clamp to close and open, or the sum of the *mold opening* and the *mold closing time*. Today's fast machines have dry-cycles in the order of 1–3 seconds. A short dry-cycle is of particular importance with fast-cycling molds. The dry-cycle also depends on the length of the clamp stroke.

*Opening time:* Usually quite fast. The *ejection* preferably should take place during this time, to reduce (or to omit completely) any *mold open time*. Occasionally, mold opening speed may need to be slowed down to suit the ejection method.

*Closing time:* Usually quite fast, except for the final approach before the mold is fully clamped up, to permit the mold protection system to operate in time before serious damage is caused to the mold.

*Mold protection:* A system which senses (at the moment of final closing the mold) whether there is foreign material (dirt, plastic pieces, products which failed to eject, etc.) between the mold halves which could cause damage to the mold. A signal from the mold protection system will cause the mold closing to stop before damage occurs and sound an alarm. It usually automatically reopens the clamp so the foreign material can be removed.

There are many types of mold protection systems, such as electric, optical, or pressure activated. Some are more sensitive than others and may not always save a mold from damage.

*Mold open (MO) time:* This is lost time. This time should be as little as possible. It can be zero.

*Mold closed (MC) time:* Time from the moment the mold has closed until it reopens. It is the sum of the following times:

*Injection time:* the time to fill the mold with plastic (usually with high injection pressure).

*Hold time:* the plastic in the cavities is held under pressure usually lower than the injection pressure to add plastic volume as the plastic shrinks within the cavity.

*Cooling time:* the time from the moment the injection (or hold) pressure is off until the mold starts opening. (This term is actually a misnomer, since the cooling is always on and starts to remove heat from the plastic as soon as the plastic enters the mold.)

*Ejection time:* the time required to eject the products from the molding area so that the mold can re-close without catching an ejected piece. Preferably, this should take place during the opening time so as to eliminate the need for additional MO time. In some molds, it is not possible or practical to eject during the mold opening, and the ejection takes place partly or solely during the MO time.

The above-defined terms can be shown on a graph describing a complete molding cycle. Figure 1.1 is a very simple graph. All motions in a mold can and should be described in such a graph. This is particularly useful where there are several motions or timed air functions within the mold and wherever there are auxiliary mechanisms, such as product removal systems (robots, chutes, guide rails, etc.). For more discussion of this type of graph, see Chapter 7, Operation Sequences.

Following are detailed explanations of the times shown in the above schematic, for better understanding of the operation of a mold and of the various features influencing the times.

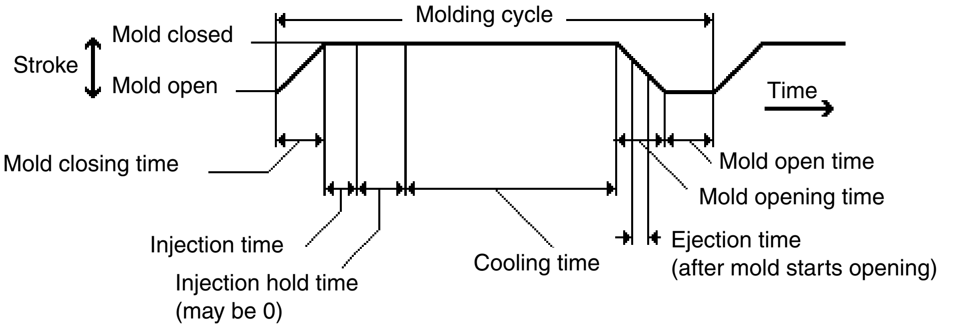


***Figure 1.1*** *Timing diagram showing complete molding cycle.*

### 1.3.1 Mold Closing and Opening Times (Dry-Cycle)

Some times are machine related. Some machines are faster than others, and speeds can be varied by settings of the machine. It is very important to understand that both the closing and opening times are more or less “wasted” times; that is, the longer they are, the smaller the productivity. Note, however, that during the mold opening stroke, the ejection can take place. The shorter the stroke, the less time is required, both for opening and for closing.

In Fig. 1.1, the entire ejection takes place during mold opening, and there should be no need for the additional MO time shown. If, however, the ejection time becomes larger and is not finished by the time the mold arrives in the mold open position, some MO time will be required.

In some cases, it may be unavoidable to delay the ejection until the mold is fully open; for example, when unloading the mold with a robot or other mechanism which is *not* driven and interlocked (synchronized) mechanically with the clamp motion. But as a rule, the mold should be designed to eject during the opening stroke without the need for MO time.

Elimination of MO time is of particular importance with thin-walled products and any other type of product which can run at very short molding cycles.

***Example:***

A container could run at a total mold closed time (injection + cooling) of 4 seconds. With a dry-cycle of 2 seconds, the total cycle would be 6 seconds, or 3,600 seconds/hour ÷ 6 seconds = 600 shots/hour. With a longer dry-cycle of 3 seconds, the cycle would be 7 seconds, or 3,600 seconds/hour ÷ 7 seconds = 514 shots/hour; obviously a large loss of production.

There is a limit of how fast a machine can cycle. A larger machine, with much larger masses to be moved, will usually run slower than a small machine. As a rule, the better a machine, the shorter the dry-cycle.

In some cases, the opening of a mold must be slowed down at the start of opening for mechanical reasons. For example, there may be a large mold separating force required, or the vacuum in the mold must be allowed to dissipate without damaging the product.

### 1.3.2 Ejection Time and Mold Opening Stroke

Ejection time is usually mold related. For example, in free fall ejection, the mold must open sufficiently far (long opening stroke) to allow all products to clear the molding area. (The larger the opening stroke, the more time is required for opening and closing, thus increasing the dry-cycle).

A long, vertical mold requires more time for the products to fall and requires more ejection time than a similar mold with a long, horizontal layout. For more information on this, see Chapter 6, Mold Layout, "Arrangement of Cavities".

### 1.3.3 Mold Open Time

Mold open (MO) time is *always* mold design related. MO time should be avoided whenever possible by trying to complete the ejection before the end of the open time.

### 1.3.4 Injection Time

The injection time depends on three factors: the machine, the mold design, and the plastic.

#### *1.3.4.1 Injection Time and the Machine*

The faster the machine can inject the required amount of plastic, the shorter the injection time. All machines are rated by the volume injected of PS (polystyrene) per second. This rate depends on the size of the hydraulic pump and motor of the machine, and the availability of accumulators to assist the injection by supplying stored high-pressure oil when the pump cannot deliver enough hydraulic oil to the injection cylinder in the required time.

#### *1.3.4.2 Injection Time and Mold Design*

Filling (and injection time) depends on:

- The *pressure drop* in the runner system from machine nozzle to, and including, the gate in the cavity. The smaller the pressure drop, the faster the mold will fill. Large runners will reduce the pressure drop, but with cold runners, they will take a long time to cool; with hot runners, the plastic inventory in the manifold becomes very large, which may affect the quality of the melt. Large gates in cold runners are unsightly and may need cutting after ejection. In hot runners, valve gates can provide large passages.
- *Number of cavities*. The more cavities, the longer it takes to fill the mold. Also, as the length of cold runners increases, they require additional plastic at every cycle. In hot runners, there is no need for more plastic once the manifold is filled, but the pressure drop increases with the runner length.
- *Product shape*. There is little the mold designer can do about product shape, but one must understand that the length of flow from gate to rim, and the wall thickness of the product are greatly influencing the speed of filling. A short flow length and a heavy wall thickness offer little resistance to filling, but a long flow and thin walls resist the filling severely, and require much higher injection pressures to push the plastic from the gate

toward the rim of the product. In addition, as the hot plastic enters the cold mold, it will immediately start to freeze and further restrict the passage between the cavity and core walls.

- *L/t Ratio* (pronounced "*L* over *t* ratio"). For thin-walled products, any ratio between flow length (*L*) and wall thickness (*t*) of greater than 200 must be treated with special attention. High injection pressures will require stronger cavity walls; also, the higher mold clamping force required to counteract the high injection pressure will affect the specific compression pressures on the parting line of the mold.

#### *1.3.4.3 Plastic*

Some plastics cannot be injected at very high speeds, to avoid degrading or burning. This must be established before starting the design by checking records of molds using similar materials or, for new plastics, with the materials suppliers.

### 1.3.5 Injection Hold Time

The injection hold time may be required to suit the product design. The method of gating selected by the designer must be suitable for injection hold pressure to be effective.

Heavy-walled products shrink considerably after injecting and will show unsightly shrink marks as the plastic solidifies where it touches the cooled cavity and core surfaces while the plastic between these colder layers is still hot and continues to shrink. This is especially noticeable at thick sections and at surfaces under ribs and hubs in the product.

The purpose of injection hold time is to maintain pressure on the plastic at the gates so that it continues filling the cavity as the product shrinks inside the cavity space. This requires relatively large, open gates which will not freeze too soon. Cold runner molds require larger than usually required edge or pin-point gates, and hot runner molds must have large, open gates or valve gates.

Thin-walled products usually have very small gates which freeze off as soon as the injection is completed; therefore, injection hold time or pressure would not be useful.

### 1.3.6 Cooling Time

The cooling time depends on many factors. Some are outside the influence of the mold designer, such as the cooling water supply of the molding plant; others are the direct result of the mold design and construction.

#### *1.3.6.1 Cooling Water Supply*

A designer should be interested in these features:

- *Water temperature*. Should be in the order of 5–10 °C (40–50 °F) or less, although certain plastics may require higher cooling water temperatures, up to 60 °C (140 °F). In injection blow molds, cooling of the core may require hot oil in the range of 100–150 °C (210–300 °F).

- *Water flow* and *water pressure*. There is no sense in having cooling water unless it is available at the mold in sufficient quantity (flow) and with sufficient pressure to maintain good circulation through the mold.
- *Clean, filtered water* which will not clog or corrode the cooling channels.
- *Connections* to the mold must be of adequate size, and laid out and installed properly, to avoid kinks causing flow restrictions.
- *Fittings* and *hoses* installed by the set-up crew must not restrict the water flow.

#### *1.3.6.2 Cooling Layout*

Proper design of the cooling layout within the mold is one of the main objectives of the mold designer. See Chapter 13, Mold Cooling.

## References

1. Menges, G., Mohren, P., Michaeli, W. (2001). *How to Make Injection Molds*, 3rd ed. Hanser Publishers, Munich.
2. Rubin, I. I. (1972). *Injection Molding: Theory and Practice.* John Wiley & Sons, New York Toronto.
3. Gastrow, H. (1992). *Injection Molds—108 proven Designs.* Hanser Publishers, Munich.
4. Stoeckhert, K. (1998). *Moldmaking Handbook for the Plastics Engineer* (2nd edition). Hanser Publishers, Munich.